Abaqus 6.12

Getting Started with Abaqus:
Keywords Edition

Getting Started with Abaqus

Keywords Edition

Legal Notices

CAUTION: This documentation is intended for qualified users who will exercise sound engineering judgment and expertise in the use of the Abaqus Software. The Abaqus Software is inherently complex, and the examples and procedures in this documentation are not intended to be exhaustive or to apply to any particular situation. Users are cautioned to satisfy themselves as to the accuracy and results of their analyses.

Dassault Systèmes and its subsidiaries, including Dassault Systèmes Simulia Corp., shall not be responsible for the accuracy or usefulness of any analysis performed using the Abaqus Software or the procedures, examples, or explanations in this documentation. Dassault Systèmes and its subsidiaries shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

The Abaqus Software is available only under license from Dassault Systèmes or its subsidiary and may be used or reproduced only in accordance with the terms of such license. This documentation is subject to the terms and conditions of either the software license agreement signed by the parties, or, absent such an agreement, the then current software license agreement to which the documentation relates.

This documentation and the software described in this documentation are subject to change without prior notice.

No part of this documentation may be reproduced or distributed in any form without prior written permission of Dassault Systèmes or its subsidiary.

The Abaqus Software is a product of Dassault Systèmes Simulia Corp., Providence, RI, USA.

© Dassault Systèmes, 2012

Abaqus, the 3DS logo, SIMULIA, CATIA, and Unified FEA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the United States and/or other countries.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information concerning trademarks, copyrights, and licenses, see the Legal Notices in the Abaqus 6.12 Installation and Licensing Guide.

Locations

SIMULIA Worldwide Headquarters	Rising Sun Mills, 166 Valley Street, Providence, RI 02909–2499, Tel: +1 401 276 4400, Fax: +1 401 276 4408, simulia.support@3ds.com, http://www.simulia.com
SIMULIA European Headquarters	Stationsplein 8-K, 6221 BT Maastricht, The Netherlands, Tel: +31 43 7999 084, Fax: +31 43 7999 306, simulia.europe.info@3ds.com

Dassault Systèmes' Centers of Simulation Excellence

United States	Fremont, CA, Tel: +1 510 794 5891, simulia.west.support@3ds.com
	West Lafayette, IN, Tel: +1 765 497 1373, simulia.central.support@3ds.com
	Northville, MI, Tel: +1 248 349 4669, simulia.greatlakes.info@3ds.com
	Woodbury, MN, Tel: +1 612 424 9044, simulia.central.support@3ds.com
	Mayfield Heights, OH, Tel: +1 216 378 1070, simulia.erie.info@3ds.com
	Mason, OH, Tel: +1 513 275 1430, simulia.central.support@3ds.com
	Warwick, RI, Tel: +1 401 739 3637, simulia.east.support@3ds.com
	Lewisville, TX, Tel: +1 972 221 6500, simulia.south.info@3ds.com
Australia	Richmond VIC, Tel: +61 3 9421 2900, simulia.au.support@3ds.com
Austria	Vienna, Tel: +43 1 22 707 200, simulia.at.info@3ds.com
Benelux	Maarssen, The Netherlands, Tel: +31 346 585 710, simulia.benelux.support@3ds.com
Canada	Toronto, ON, Tel: +1 416 402 2219, simulia.greatlakes.info@3ds.com
China	Beijing, P. R. China, Tel: +8610 6536 2288, simulia.cn.support@3ds.com
	Shanghai, P. R. China, Tel: +8621 3856 8000, simulia.cn.support@3ds.com
Finland	Espoo, Tel: +358 40 902 2973, simulia.nordic.info@3ds.com
France	Velizy Villacoublay Cedex, Tel: +33 1 61 62 72 72, simulia.fr.support@3ds.com
Germany	Aachen, Tel: +49 241 474 01 0, simulia.de.info@3ds.com
	Munich, Tel: +49 89 543 48 77 0, simulia.de.info@3ds.com
India	Chennai, Tamil Nadu, Tel: +91 44 43443000, simulia.in.info@3ds.com
Italy	Lainate MI, Tel: +39 02 3343061, simulia.ity.info@3ds.com
Japan	Tokyo, Tel: +81 3 5442 6302, simulia.jp.support@3ds.com
	Osaka, Tel: +81 6 7730 2703, simulia.jp.support@3ds.com
Korea	Mapo-Gu, Seoul, Tel: +82 2 785 6707/8, simulia.kr.info@3ds.com
Latin America	Puerto Madero, Buenos Aires, Tel: +54 11 4312 8700, Horacio.Burbridge@3ds.com
Scandinavia	Stockholm, Sweden, Tel: +46 8 68430450, simulia.nordic.info@3ds.com
United Kingdom	Warrington, Tel: +44 1925 830900, simulia.uk.info@3ds.com

Authorized Support Centers

Argentina	SMARTtech Sudamerica SRL, Buenos Aires, Tel: +54 11 4717 2717
	KB Engineering, Buenos Aires, Tel: +54 11 4326 7542
	Solaer Ingeniería, Buenos Aires, Tel: +54 221 489 1738
Brazil	SMARTtech Mecânica, Sao Paulo-SP, Tel: +55 11 3168 3388
Czech & Slovak Republics	Synerma s. r. o., Psáry, Prague-West, Tel: +420 603 145 769, abaqus@synerma.cz
Greece	3 Dimensional Data Systems, Crete, Tel: +30 2821040012, support@3dds.gr
Israel	ADCOM, Givataim, Tel: +972 3 7325311, shmulik.keidar@adcomsim.co.il
Malaysia	WorleyParsons Services Sdn. Bhd., Kuala Lumpur, Tel: +603 2039 9000, abaqus.my@worleyparsons.com
Mexico	Kimeca.NET SA de CV, Mexico, Tel: +52 55 2459 2635
New Zealand	Matrix Applied Computing Ltd., Auckland, Tel: +64 9 623 1223, abaqus-tech@matrix.co.nz
Poland	BudSoft Sp. z o.o., Poznań, Tel: +48 61 8508 466, info@budsoft.com.pl
Russia, Belarus & Ukraine	TESIS Ltd., Moscow, Tel: +7 495 612 44 22, info@tesis.com.ru
Singapore	WorleyParsons Pte Ltd., Singapore, Tel: +65 6735 8444, abaqus.sg@worleyparsons.com
South Africa	Finite Element Analysis Services (Pty) Ltd., Parklands, Tel: +27 21 556 6462, feas@feas.co.za
Spain & Portugal	Principia Ingenieros Consultores, S.A., Madrid, Tel: +34 91 209 1482, simulia@principia.es

Taiwan	Simutech Solution Corporation, Taipei, R.O.C., Tel: +886 2 2507 9550, camilla@simutech.com.tw
Thailand	WorleyParsons Pte Ltd., Singapore, Tel: +65 6735 8444, abaqus.sg@worleyparsons.com
Turkey	A-Ztech Ltd., Istanbul, Tel: +90 216 361 8850, info@a-ztech.com.tr

Complete contact information is available at http://www.simulia.com/locations/locations.html.

Contents

1. Introduction

The Abaqus products	1.1
Getting started with Abaqus	1.2
Abaqus documentation	1.3
Getting help	1.4
Support	1.5
A quick review of the finite element method	1.6

2. Abaqus Basics

Components of an Abaqus analysis model	2.1
Format of the input file	2.2
Example: creating a model of an overhead hoist	2.3
Comparison of implicit and explicit procedures	2.4
Summary	2.5

3. Finite Elements and Rigid Bodies

Finite elements	3.1
Rigid bodies	3.2
Summary	3.3

4. Using Continuum Elements

Element formulation and integration	4.1
Selecting continuum elements	4.2
Example: connecting lug	4.3
Mesh convergence	4.4
Related Abaqus examples	4.5
Suggested reading	4.6
Summary	4.7

5. Using Shell Elements

Element geometry	5.1
Shell formulation – thick or thin	5.2
Shell material directions	5.3
Selecting shell elements	5.4
Example: skew plate	5.5
Related Abaqus examples	5.6

Suggested reading 5.7
Summary 5.8

6. Using Beam Elements

Beam cross-section geometry 6.1
Formulation and integration 6.2
Selecting beam elements 6.3
Example: cargo crane 6.4
Related Abaqus examples 6.5
Suggested reading 6.6
Summary 6.7

7. Linear Dynamics

Introduction 7.1
Damping 7.2
Element selection 7.3
Mesh design for dynamics 7.4
Example: cargo crane under dynamic loading 7.5
Effect of the number of modes 7.6
Effect of damping 7.7
Comparison with direct time integration 7.8
Other dynamic procedures 7.9
Related Abaqus examples 7.10
Suggested reading 7.11
Summary 7.12

8. Nonlinearity

Sources of nonlinearity 8.1
The solution of nonlinear problems 8.2
Including nonlinearity in an Abaqus analysis 8.3
Example: nonlinear skew plate 8.4
Related Abaqus examples 8.5
Suggested reading 8.6
Summary 8.7

9. Nonlinear Explicit Dynamics

Types of problems suited for Abaqus/Explicit 9.1
Explicit dynamic finite element methods 9.2
Automatic time incrementation and stability 9.3
Example: stress wave propagation in a bar 9.4
Damping of dynamic oscillations 9.5

Energy balance 9.6
Summary 9.7

10. Materials

Defining materials in Abaqus 10.1
Plasticity in ductile metals 10.2
Selecting elements for elastic-plastic problems 10.3
Example: connecting lug with plasticity 10.4
Example: blast loading on a stiffened plate 10.5
Hyperelasticity 10.6
Example: axisymmetric mount 10.7
Mesh design for large distortions 10.8
Techniques for reducing volumetric locking 10.9
Related Abaqus examples 10.10
Suggested reading 10.11
Summary 10.12

11. Multiple Step Analysis

General analysis procedures 11.1
Linear perturbation analysis 11.2
Example: vibration of a piping system 11.3
Restart analysis 11.4
Example: restarting the pipe vibration analysis 11.5
Related Abaqus examples 11.6
Summary 11.7

12. Contact

Overview of contact capabilities in Abaqus 12.1
Interaction between surfaces 12.2
Defining contact in Abaqus/Standard 12.3
Modeling issues for rigid surfaces in Abaqus/Standard 12.4
Abaqus/Standard 2-D example: forming a channel 12.5
General contact in Abaqus/Standard 12.6
Abaqus/Standard 3-D example: shearing of a lap joint 12.7
Defining contact in Abaqus/Explicit 12.8
Modeling considerations in Abaqus/Explicit 12.9
Abaqus/Explicit example: circuit board drop test 12.10
Compatibility between Abaqus/Standard and Abaqus/Explicit 12.11
Related Abaqus examples 12.12
Suggested reading 12.13
Summary 12.14

13. Quasi-Static Analysis with Abaqus/Explicit

Analogy for explicit dynamics	13.1
Loading rates	13.2
Mass scaling	13.3
Energy balance	13.4
Example: forming a channel in Abaqus/Explicit	13.5
Summary	13.6

A. Example Files

Overhead hoist frame	A.1
Connecting lug	A.2
Skew plate	A.3
Cargo crane	A.4
Cargo crane – dynamic loading	A.5
Nonlinear skew plate	A.6
Stress wave propagation in a bar	A.7
Connecting lug with plasticity	A.8
Blast loading on a stiffened plate	A.9
Axisymmetric mount	A.10
Test fit of hyperelastic material data	A.11
Vibration of a piping system	A.12
Forming a channel with Abaqus/Standard	A.13
Shearing of a lap joint	A.14
Circuit board drop test	A.15
Forming a channel with Abaqus/Explicit	A.16

1. Introduction

Abaqus is a suite of powerful engineering simulation programs, based on the finite element method, that can solve problems ranging from relatively simple linear analyses to the most challenging nonlinear simulations. Abaqus contains an extensive library of elements that can model virtually any geometry. It has an equally extensive list of material models that can simulate the behavior of most typical engineering materials including metals, rubber, polymers, composites, reinforced concrete, crushable and resilient foams, and geotechnical materials such as soils and rock. Designed as a general-purpose simulation tool, Abaqus can be used to study more than just structural (stress/displacement) problems. It can simulate problems in such diverse areas as heat transfer, mass diffusion, thermal management of electrical components (coupled thermal-electrical analyses), acoustics, soil mechanics (coupled pore fluid-stress analyses), piezoelectric analysis, electromagnetic analysis, and fluid dynamics.

Abaqus offers a wide range of capabilities for simulation of linear and nonlinear applications. Problems with multiple components are modeled by associating the geometry defining each component with the appropriate material models and specifying component interactions. In a nonlinear analysis Abaqus automatically chooses appropriate load increments and convergence tolerances and continually adjusts them during the analysis to ensure that an accurate solution is obtained efficiently.

1.1 The Abaqus products

Abaqus consists of three main analysis products—Abaqus/Standard, Abaqus/Explicit, and Abaqus/CFD. Several add-on analysis options are available to further extend the capabilities of Abaqus/Standard and Abaqus/Explicit. The Abaqus/Aqua option works with Abaqus/Standard and Abaqus/Explicit. The Abaqus/Design and Abaqus/AMS options work with Abaqus/Standard. Abaqus/Foundation is an optional subset of Abaqus/Standard. Abaqus/CAE is the complete Abaqus environment that includes capabilities for creating Abaqus models, interactively submitting and monitoring Abaqus jobs, and evaluating results. Abaqus/Viewer is a subset of Abaqus/CAE that includes just the postprocessing functionality. In addition, the Abaqus Interface for Moldflow and the Abaqus Interface for MSC.ADAMS are interfaces to Moldflow and ADAMS/Flex, respectively. Abaqus also provides translators that convert geometry from third-party CAD systems to models for Abaqus/CAE, convert entities from third-party preprocessors to input for Abaqus analyses, and that convert output from Abaqus analyses to entities for third-party postprocessors. The relationship between these products is shown in Figure 1–1.

Abaqus/Standard

Abaqus/Standard is a general-purpose analysis product that can solve a wide range of linear and nonlinear problems involving the static, dynamic, thermal, electrical, and electromagnetic response of components. This product is discussed in detail in this guide. Abaqus/Standard solves a system of equations implicitly at each solution "increment." In contrast, Abaqus/Explicit marches a solution

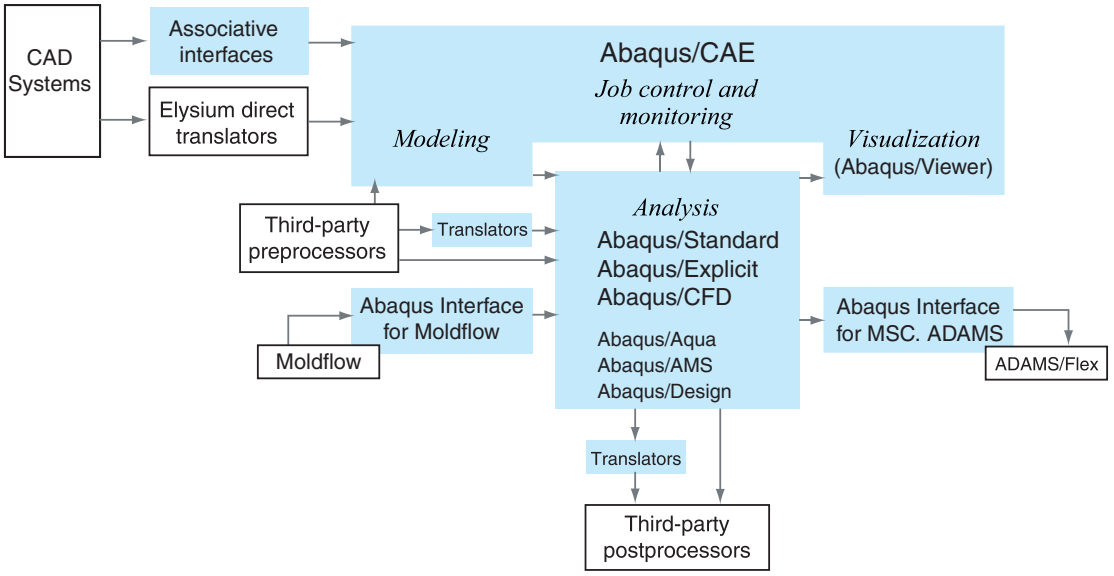


Figure 1–1 Abaqus products.

forward through time in small time increments without solving a coupled system of equations at each increment (or even forming a global stiffness matrix).

Abaqus/Explicit

Abaqus/Explicit is a special-purpose analysis product that uses an explicit dynamic finite element formulation. It is suitable for modeling brief, transient dynamic events, such as impact and blast problems, and is also very efficient for highly nonlinear problems involving changing contact conditions, such as forming simulations. Abaqus/Explicit is discussed in detail in this guide.

Abaqus/CFD

Abaqus/CFD is a computational fluid dynamics analysis product. It can solve a broad class of incompressible flow problems including laminar and turbulent flow, thermal convective flow, and deforming mesh problems. Abaqus/CFD is discussed in this guide.

Abaqus/CAE

Abaqus/CAE (Complete Abaqus Environment) is an interactive, graphical environment for Abaqus. It allows models to be created quickly and easily by producing or importing the geometry of the structure to be analyzed and decomposing the geometry into meshable regions. Physical and material properties can be assigned to the geometry, together with loads and boundary conditions. Abaqus/CAE contains very powerful options to mesh the geometry and to verify the resulting

analysis model. Once the model is complete, Abaqus/CAE can submit, monitor, and control the analysis jobs. The Visualization module can then be used to interpret the results.

Abaqus/Viewer, which is a subset of Abaqus/CAE that contains only the postprocessing capabilities of the Visualization module, is discussed in this guide. The other Abaqus/CAE modules are not discussed in this guide.

Abaqus/Aqua

Abaqus/Aqua is a set of optional capabilities that can be added to Abaqus/Standard and Abaqus/Explicit. It is intended for the simulation of offshore structures, such as oil platforms. Some of the optional capabilities include the effects of wave and wind loading and buoyancy. Abaqus/Aqua is not discussed in this guide.

Abaqus/Design

Abaqus/Design is a set of optional capabilities that can be added to Abaqus/Standard to perform design sensitivity calculations. Abaqus/Design is not discussed in this guide.

Abaqus/AMS

Abaqus/AMS is an optional capability that can be added to Abaqus/Standard. It uses the automatic multi-level substructuring (AMS) eigensolver during a natural frequency extraction. Abaqus/AMS is not discussed in this guide.

Abaqus/Foundation

Abaqus/Foundation offers more efficient access to the linear static and dynamic analysis functionality in Abaqus/Standard. Abaqus/Foundation is not discussed in this guide.

Abaqus Interface for Moldflow

The Abaqus Interface for Moldflow translates finite element model information from a Moldflow analysis to write a partial Abaqus input file. The Abaqus Interface for Moldflow is not discussed in this guide.

Abaqus Interface for MSC.ADAMS

The Abaqus Interface for MSC.ADAMS allows Abaqus finite element models to be included as flexible components within the MSC.ADAMS family of products. The interface is based on the component mode synthesis formulation of ADAMS/Flex. The Abaqus Interface for MSC.ADAMS is not discussed in this guide.

Geometry translators

Abaqus provides the following translators for converting geometry from third-party CAD systems to parts and assemblies for Abaqus/CAE:

- The SIMULIA Associative Interface for Abaqus/CAE creates a link between CATIA V6 and Abaqus/CAE that allows you to transfer model data and propagate design changes from CATIA V6 to Abaqus/CAE.

- The CATIA V5 Associative Interface creates a link between CATIA V5 and Abaqus/CAE that allows you to transfer model data and propagate design changes from CATIA V5 to Abaqus/CAE.
- The SolidWorks Associative Interface creates a link between SolidWorks and Abaqus/CAE that allows you to transfer model data and propagate design changes from SolidWorks to Abaqus/CAE.
- The Pro/ENGINEER Associative Interface creates a link between Pro/ENGINEER and Abaqus/CAE that allows you to transfer model data and propagate design changes between Pro/ENGINEER and Abaqus/CAE.
- The Geometry Translator for CATIA V4 allows you to import the geometry of CATIA V4-format parts and assemblies directly into Abaqus/CAE.
- The Geometry Translator for Parasolid allows you to import the geometry of Parasolid-format parts and assemblies directly into Abaqus/CAE.

In addition, the Abaqus/CAE Associative Interface for NX creates a link between NX and Abaqus/CAE that allows you to transfer model data and propagate design changes between NX and Abaqus/CAE. The Abaqus/CAE Associative Interface for NX can be purchased and downloaded from Elysium Inc. (www.elysiuminc.com).

The geometry translators are not discussed in this guide.

Translator utilities

Abaqus provides the following translators for converting entities from third-party preprocessors to input for Abaqus analyses or for converting output from Abaqus analyses to entities for third-party postprocessors:

- **abaqus fromansys** translates an ANSYS input file to an Abaqus input file.
- **abaqus fromdyna** translates an LS-DYNA keyword file to an Abaqus input file.
- **abaqus fromnastran** translates a Nastran bulk data file to an Abaqus input file.
- **abaqus frompamcrash** translates a PAM-CRASH input file into an Abaqus input file.
- **abaqus fromradioss** translates a RADIOSS input file into an Abaqus input file.
- **abaqus tonastran** translates an Abaqus input file to Nastran bulk data file format.
- **abaqus toOutput2** translates an Abaqus output database file to the Nastran Output2 file format.
- **abaqus tozaero** enables the exchange of aeroelastic data between Abaqus and ZAERO.

The translator utilities are not discussed in this guide.

1.2 Getting started with Abaqus

This guide is an introductory text designed to give new users guidance in analyzing solid, shell, beam, and truss models with Abaqus/Standard and Abaqus/Explicit, and viewing the results in Abaqus/Viewer

or another postprocessor. You do not need any previous knowledge of Abaqus to benefit from this guide, although some previous exposure to the finite element method is recommended. If you are already familiar with the Abaqus solver products (Abaqus/Standard or Abaqus/Explicit) but would like an introduction to the Abaqus/CAE interface, refer to the Getting Started with Abaqus: Interactive Edition manual.

This document covers primarily stress/displacement simulations, concentrating on both linear and nonlinear static analyses as well as dynamic analyses. An introduction to CFD analysis and modeling fluid-structure interaction is also included. Other types of simulations, such as heat transfer and mass diffusion, are not covered.

1.2.1 How to use this guide

Each of the chapters in this guide introduces one or more topics relevant to using Abaqus/Standard and Abaqus/Explicit. Throughout the manual the term Abaqus is used to refer collectively to both Abaqus/Standard and Abaqus/Explicit; the individual product names are used when information applies to only one product. Most chapters contain a short discussion of the topic or topics being considered and one or two tutorial examples. You should work through the examples carefully since they contain a great deal of practical advice on using Abaqus.

The capabilities of Abaqus/Standard and Abaqus/Explicit are introduced gradually in these examples. You may create input files using a text editor; however, using an interactive pre-processor facilitates model creation for these examples. Full versions of the input files that you create in each example are in Appendix A, “Example Files.” If you have access to Abaqus/CAE, you can use the companion manual, Getting Started with Abaqus: Interactive Edition, to perform all preprocessing and analysis steps using detailed Abaqus/CAE tutorials.

This chapter is a short introduction to Abaqus and this guide. Chapter 2, “Abaqus Basics,” which is centered around a simple example, covers the basics of using Abaqus. By the end of Chapter 2, “Abaqus Basics,” you will know the fundamentals of how to prepare a model for an Abaqus simulation, check the data, run the analysis job, and view the results.

Chapter 3, “Finite Elements and Rigid Bodies,” presents an overview of the main element families available in Abaqus. The use of continuum (solid) elements, shell elements, and beam elements is discussed in Chapter 4, “Using Continuum Elements”; Chapter 5, “Using Shell Elements”; and Chapter 6, “Using Beam Elements”; respectively.

Linear dynamic analyses are discussed in Chapter 7, “Linear Dynamics.” Chapter 8, “Nonlinearity,” introduces the concept of nonlinearity in general, and geometric nonlinearity in particular, and contains the first nonlinear Abaqus simulation. Nonlinear dynamic analyses are discussed in Chapter 9, “Nonlinear Explicit Dynamics,” and material nonlinearity is introduced in Chapter 10, “Materials.” Chapter 11, “Multiple Step Analysis,” introduces the concept of multistep simulations, and Chapter 12, “Contact,” discusses the many issues that arise in contact analyses. Using Abaqus/Explicit to solve quasi-static problems is presented in Chapter 13, “Quasi-Static Analysis with Abaqus/Explicit.” The illustrative example is a sheet metal forming simulation, which requires importing between Abaqus/Explicit and Abaqus/Standard to perform the forming and springback analyses efficiently.

1.2.2 Conventions used in this guide

This manual adheres to the following conventions:

Typographical conventions

Different text styles are used in the tutorial examples to indicate specific actions or identify items.

- Input in **`COURIER FONT`** should be typed into Abaqus/Viewer or your computer exactly as shown. For example,

  ```
  abaqus viewer
  ```

  would be typed on your computer to run Abaqus/Viewer.
- Menu selections, tabs within dialog boxes, and labels of items on the screen in Abaqus/Viewer are indicated in **bold**:

  View→Graphics Options
  Contour Plot Options

View orientation triad

By default, Abaqus/Viewer uses the alphabetical option, *x-y-z*, for labeling the view orientation triad. In general, this manual adopts the numerical option, 1-2-3, to permit direct correspondence with degree of freedom and output labeling.

1.2.3 Basic mouse actions

Figure 1–2 shows the mouse button orientation for a left-handed and a right-handed 3-button mouse.

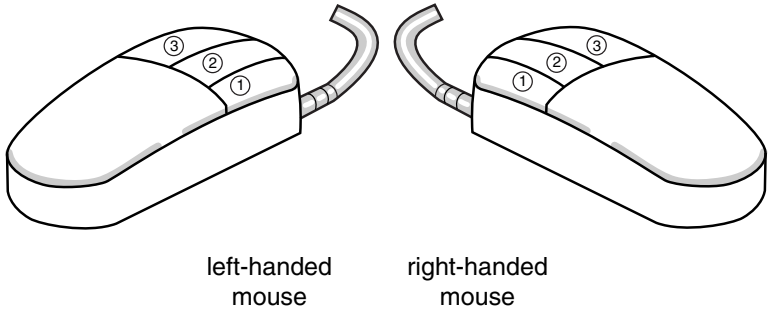


Figure 1–2 Mouse buttons.

The following terms describe actions you perform using the mouse:

Click

Press and quickly release the mouse button. Unless otherwise specified, the instruction "click" means that you should click mouse button 1.

Drag

Press and hold down mouse button 1 while moving the mouse.

Point

Move the mouse until the cursor is over the desired item.

Select

Point to an item and then click mouse button 1.

[Shift]+Click

Press and hold the [Shift] key, click mouse button 1, and then release the [Shift] key.

[Ctrl]+Click

Press and hold the [Ctrl] key, click mouse button 1, and then release the [Ctrl] key.

Abaqus/Viewer is designed for use with a 3-button mouse. Accordingly, this manual refers to mouse buttons 1, 2, and 3 as shown in Figure 1–2. However, you can use Abaqus/Viewer with a 2-button mouse as follows:

- The two mouse buttons are equivalent to mouse buttons 1 and 3 on a 3-button mouse.
- Pressing both mouse buttons simultaneously is equivalent to pressing mouse button 2 on a 3-button mouse.

  Tip: You are instructed to click mouse button 2 in procedures throughout this manual. Make sure that you configure mouse button 2 (or the wheel button) to act as a middle button click.

1.3 Abaqus documentation

The documentation for Abaqus is extensive and complete. The following documentation and publications are available from SIMULIA through the Abaqus online HTML documentation and in PDF format. For more information on accessing the online HTML manuals, refer to the discussion of execution procedures in the Abaqus Analysis User's Manual. For more information on printing the manuals, refer to "Printing from a PDF book," Section 5.3 of Using Abaqus Online Documentation.

Abaqus Analysis User's Manual

This manual contains a complete description of the elements, material models, procedures, input specifications, etc. It is the basic manual for Abaqus/Standard, Abaqus/Explicit, and Abaqus/CFD; and it provides both input file usage and Abaqus/CAE usage information. This guide regularly

refers to the Abaqus Analysis User's Manual, so you should have it available as you work through the examples.

Abaqus/CAE User's Manual

This manual includes detailed descriptions of how to use Abaqus/CAE for model generation, analysis, and results evaluation and visualization. Abaqus/Viewer users should refer to the information on the Visualization module in this manual.

Using Abaqus Online Documentation

This manual contains instructions for navigating, viewing, and searching the Abaqus HTML and PDF documentation. In addition, this manual explains how to use the PDF documentation to produce a high quality printed copy and how to use the icon in all PDF books except the Abaqus Scripting Reference Manual and the Abaqus GUI Toolkit Reference Manual to print a selected section of a book.

Other Abaqus documentation:

Abaqus Example Problems Manual

This manual contains detailed examples designed to illustrate the approaches and decisions needed to perform meaningful linear and nonlinear analysis. Many of the examples are worked with several different element types, mesh densities, and other variations. Typical cases are large motion of an elastic-plastic pipe hitting a rigid wall; inelastic buckling collapse of a thin-walled elbow; explosive loading of an elastic, viscoplastic thin ring; consolidation under a footing; buckling of a composite shell with a hole; and deep drawing of a metal sheet. It is generally useful to look for relevant examples in this manual and to review them when embarking on a new class of problem.

When you want to use a feature that you have not used before, you should look up one or more examples that use that feature. Then, use the example to familiarize yourself with the correct usage of the capability. To find an example that uses a certain feature, search the online documentation or use the **abaqus findkeyword** utility (see "Querying the keyword/problem database," Section 3.2.13 of the Abaqus Analysis User's Manual, for more information).

All the input files associated with the examples are provided as part of the Abaqus installation. The **abaqus fetch** utility is used to extract sample Abaqus input files from the compressed archive files provided with the release (see "Fetching sample input files," Section 3.2.14 of the Abaqus Analysis User's Manual, for more information). You can fetch any of the example files so that you can run the simulations yourself and review the results. You can also access the input files through the hyperlinks in the Abaqus Example Problems Manual.

Abaqus Benchmarks Manual

This manual contains benchmark problems and analyses used to evaluate the performance of Abaqus; the tests are multiple element tests of simple geometries or simplified versions of real problems. The NAFEMS benchmark problems are included in this manual.

Abaqus Verification Manual

This manual contains basic test cases, providing verification of each individual program feature (procedures, output options, MPCs, etc.) against exact calculations and other published results. It may be useful to run these problems when learning to use a new capability. In addition, the supplied input data files provide good starting points to check the behavior of elements, materials, etc.

Abaqus Theory Manual

This manual contains detailed, precise discussions of all theoretical aspects of Abaqus. It is written to be understood by users with an engineering background.

Abaqus Keywords Reference Manual

This manual contains a complete description of all the input options that are available in Abaqus/Standard, Abaqus/Explicit, and Abaqus/CFD.

Abaqus User Subroutines Reference Manual

This manual contains a complete description of all the user subroutines available for use in Abaqus analyses. It also discusses the utility routines that can be used when writing user subroutines.

Abaqus Glossary

This manual defines technical terms as they apply to the Abaqus Unified FEA Product Suite.

Abaqus Release Notes

This manual contains brief descriptions of the new features available in the latest release of the Abaqus product line.

Abaqus Installation and Licensing Guide

This manual describes how to install Abaqus and how to configure the installation for particular circumstances. Some of this information, of most relevance to users, is also provided in the Abaqus Analysis User's Manual.

In addition to the documentation listed above, the following manuals are available for Abaqus interfaces and custom programming techniques not discussed in this guide:

- Abaqus Interface for Moldflow User's Manual
- Abaqus Interface for MSC.ADAMS User's Manual
- Abaqus Scripting User's Manual
- Abaqus Scripting Reference Manual
- Abaqus GUI Toolkit User's Manual
- Abaqus GUI Toolkit Reference Manual

SIMULIA also provides documentation for all of the geometry translators described in "The Abaqus products," Section 1.1.

Additional publications available from SIMULIA:

Quality Assurance Plan

This document describes the QA procedures followed by SIMULIA. It is a controlled document, provided to customers who subscribe to either the Nuclear QA Program or the Quality Monitoring Service.

Lecture Notes

These notes are available on many topics to which Abaqus is applied. They are used in the technical seminars that are presented to help users improve their understanding and usage of Abaqus. While not intended as stand-alone tutorial material, they are sufficiently comprehensive that they can usually be used in that mode.

Abaqus online resources

SIMULIA has a home page on the World Wide Web (www.simulia.com), containing a variety of useful information about the Abaqus suite of programs, including:

- Frequently asked questions
- Abaqus systems information and machine requirements
- Benchmark timing documents
- Error status reports
- Abaqus documentation price list
- Training course schedule
- Newsletters

1.4 Getting help

You may want to read additional information about Abaqus/Viewer features at various points during the tutorials. The context-sensitive help system allows you to locate relevant information quickly and easily. Context-sensitive help is available for every item in the main window and in all dialog boxes.

Note:

- On Windows platforms, the help system uses your default web browser to display the online documentation.
- On UNIX and Linux platforms, the help system searches the system path for Firefox. If the help system cannot find Firefox, an error is displayed.

  The **browser_type** and **browser_path** variables can be set in the environment file to modify this behavior. For more information, see "System customization parameters," Section 4.1.4 of the Abaqus Installation and Licensing Guide.

To obtain context-sensitive help:

1. From the main menu bar, select **Help→On Context**.

   Tip: You can also click the help tool to access context-sensitive help.

   The cursor changes to a question mark.

2. Click any part of the main window except its frame.

   A help window appears in your browser window. The help window displays information about the item you selected.

3. Scroll to the bottom of the help window.

   At the bottom of the window, a list of blue, underlined items appears. These items are links to the Abaqus/CAE User's Manual, which includes all Abaqus/Viewer help topics.

4. Click any one of the items.

   A book window appears in your default web browser. The window is arranged into four frames as follows:

   - The Abaqus/CAE User's Manual appears in a text frame on the right side of the window. The manual is turned to the item that you selected.
   - An expandable table of contents is available on the lower left side of the window for easy navigation throughout the book.
   - The table of contents control tools in the upper left frame allow you to vary the level of detail displayed in the table of contents frame or to change the size of the frame. Click to expand several levels in the table of contents of an online book. Click to collapse all expanded sections in the table of contents. Click and , respectively, to widen or narrow the table of contents frame.
   - The navigation frame at the top of the book window allows you to select another book from the entire Abaqus documentation collection. The navigation frame also allows you to search the entire manual.

5. Click any item in the table of contents.

   The text frame changes to reflect the item you selected.

6. Click the icon to the left of a topic heading to expand it.

   The headings of the subtopics appear under the topic heading, and the sign changes to , indicating that the section is expanded. If appears beside a subsection, there are no further levels within that section to expand. To collapse an expanded section of the table of contents, click next to the topic heading.

7. In the search panel in the navigation frame, type any word that appears in the text frame on the right and click **Search**.

   When the search is complete, the table of contents frame displays the number of hits next to each topic heading and all hits become highlighted in the text frame. Click **Next Match** or **Previous Match** in the navigation frame to move through the document from one hit to the next.

   You can enter a single word or a phrase in the search panel, and you can use the [*] character as a wildcard. For detailed instructions on using the search capabilities of the online documentation, see Using Abaqus Online Documentation.

8. Close the web browser windows.

1.5 Support

Both technical engineering support (for problems with creating a model or performing an analysis) and systems support (for installation, licensing, and hardware-related problems) for Abaqus are offered through a network of local support offices. Regional contact information is listed in the front of each Abaqus manual and is accessible from the **Locations** page at www.simulia.com. Support is also available from the **Support** page at www.simulia.com. When contacting your local support office, please specify whether you would like technical support (you have encountered problems performing an Abaqus analysis) or systems support (Abaqus will not install correctly, licensing does not work correctly, or other hardware-related issues have arisen).

We welcome any suggestions for improvements to Abaqus software, the support program, or documentation. We will ensure that any enhancement requests you make are considered for future releases. If you wish to make a suggestion about the service or products provided by SIMULIA, refer to www.simulia.com. Complaints should be addressed by contacting your local office or through www.simulia.com by visiting the **Quality Assurance** section of the **Support** page.

1.5.1 Technical support

SIMULIA technical support engineers can assist in clarifying Abaqus features and checking errors by giving both general information on using Abaqus and information on its application to specific analyses. If you have concerns about an analysis, we suggest that you contact us at an early stage, since it is usually easier to solve problems at the beginning of a project rather than trying to correct an analysis at the end.

Please have the following information ready before calling the technical support hotline, and include it in any written contacts:

- The release of Abaqus that are you using.
  - The release numbers for Abaqus/Standard, Abaqus/Explicit, and Abaqus/CFD are given at the top of the data (`.dat`) file.

  - The release numbers for the Abaqus Interface for Moldflow and the Abaqus Interface for MSC.ADAMS are output to the screen.

- The type of computer on which you are running Abaqus.
- The symptoms of any problems, including the exact error messages, if any.
- Workarounds or tests that you have already tried.

For support about a specific problem, any available Abaqus output files may be helpful in answering questions that the support engineer may ask you.

The support engineer will try to diagnose your problem from the model description and a description of the difficulties you are having. Frequently, the support engineer will need model sketches, which can be e-mailed, faxed, or sent in the mail. Plots of the final results or the results near the point that the analysis terminated may also be needed to understand what may have caused the problem.

If the support engineer cannot diagnose your problem from this information, you may be asked to supply the input data. The data can be attached to a support incident in the online system. It can also be sent by means of e-mail, ftp, CD, or DVD. Please check the **Support Overview** page at www.simulia.com for the media formats that are currently accepted.

All support incidents are tracked. This tracking enables you (as well as the support engineer) to monitor the progress of a particular problem and to check that we are resolving support issues efficiently. You must register with the system to check on a support issue. If you contact us by means outside the system to discuss an existing support problem and you know the incident or support request number, please mention it so that we can consult the database to see what the latest action has been.

1.5.2 Systems support

Abaqus systems support engineers can help you resolve issues related to the installation and running of Abaqus, including licensing difficulties, that are not covered by technical support.

You should install Abaqus by carefully following the instructions in the Abaqus Installation and Licensing Guide. If you encounter problems with the installation or licensing, first review the instructions in the Abaqus Installation and Licensing Guide to ensure that they have been followed correctly. If this method does not resolve the problems, search for known installation problems in the Dassault Systèmes Knowledge Base at www.3ds.com/support/knowledge-base or the SIMULIA Online Support System, which is accessible through the **My Support** page at www.simulia.com. If this method does not address your situation, please contact your local support office; a list of local support offices is available from the **Locations** page at www.simulia.com. Send whatever information is available to define the problem: error messages from an aborted analysis or a detailed explanation of the problems encountered. Whenever possible, please send the output from the **abaqus info=support** command.

1.5.3 Support for academic institutions

Under the terms of the Academic License Agreement, we do not provide support to users at academic institutions unless the institution has also purchased technical support. Please contact us for more information.

1.6 A quick review of the finite element method

This section reviews the basics of the finite element method. The first step of any finite element simulation is to *discretize* the actual geometry of the structure using a collection of *finite elements*. Each finite element represents a discrete portion of the physical structure. The finite elements are joined by shared *nodes*. The collection of nodes and finite elements is called the *mesh*. The number of elements per unit of length, area, or in a mesh is referred to as the *mesh density*. In a stress analysis the displacements of the nodes are the fundamental variables that Abaqus calculates. Once the nodal displacements are known, the stresses and strains in each finite element can be determined easily.

1.6.1 Obtaining nodal displacements using implicit methods

A simple example of a truss, constrained at one end and loaded at the other end as shown in Figure 1–3, is used to introduce some terms and conventions used in this document.

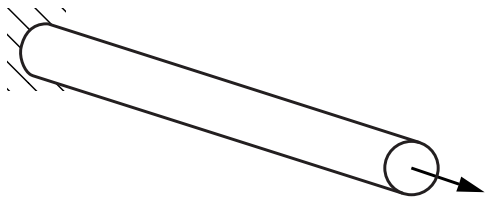


Figure 1–3 Truss problem.

The objective of the analysis is to find the displacement of the free end of the truss, the stress in the truss, and the reaction force at the constrained end of the truss.

In this case the rod shown in Figure 1–3 will be modeled with two truss elements. In Abaqus truss elements can carry axial loads only. The discretized model is shown in Figure 1–4 together with the node and element labels.

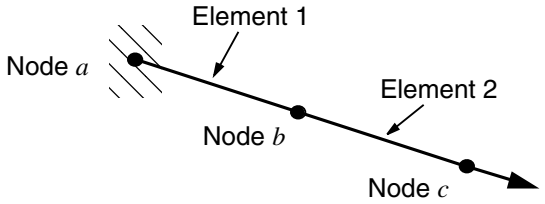


Figure 1–4 Discretized model of the truss problem.

Free-body diagrams for each node in the model are shown in Figure 1–5. In general each node will carry an external load applied to the model, $\boldsymbol{P}$, and internal loads, $\boldsymbol{I}$, caused by stresses in the elements attached to that node. For a model to be in static equilibrium, the net force acting on each node must be zero; i.e., the internal and external loads at each node must balance each other. For node $\boldsymbol{a}$ this equilibrium equation can be obtained as follows.

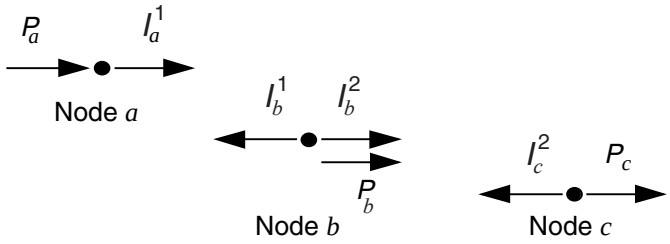


Figure 1–5 Free-body diagram for each node.

Assuming that the change in length of the rod is small, the strain in element 1 is given by

$$\varepsilon_{11} = \frac{u^b - u^a}{L},$$

where u^a and u^b are the displacements at nodes $\boldsymbol{a}$ and $\boldsymbol{b}$, respectively, and $\boldsymbol{L}$ is the original length of the element.

Assuming that the material is elastic, the stress in the rod is given by the strain multiplied by the Young's modulus, $\boldsymbol{E}$:

$$\sigma_{11} = E\varepsilon_{11}.$$

The axial force acting on the end node is equivalent to the stress in the rod multiplied by its cross-sectional area, ***A***. Thus, a relationship between internal force, material properties, and displacements is obtained:

$$I_a^1 = \sigma_{11} A = E\varepsilon_{11} A = \frac{EA}{L}(u^b - u^a).$$

Equilibrium at node ***a*** can, therefore, be written as

$$P_a + \frac{EA}{L}(u^b - u^a) = 0.$$

Equilibrium at node ***b*** must take into account the internal forces acting from both elements joined at that node. The internal force from element 1 is now acting in the opposite direction and so becomes negative. The resulting equation is

$$P_b - \frac{EA}{L}(u^b - u^a) + \frac{EA}{L}(u^c - u^b) = 0.$$

For node *c* the equilibrium equation is

$$P_c - \frac{EA}{L}(u^c - u^b) = 0.$$

For implicit methods, the equilibrium equations need to be solved simultaneously to obtain the displacements of all the nodes. This requirement is best achieved by matrix techniques; therefore, write the internal and external force contributions as matrices. If the properties and dimensions of the two elements are the same, the equilibrium equations can be simplified as follows:

$$\begin{Bmatrix} P_a \\ P_b \\ P_c \end{Bmatrix} - \left(\frac{EA}{L}\right) \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} u^a \\ u^b \\ u^c \end{Bmatrix} = 0.$$

In general, it may be that the element stiffnesses, the EA/L terms, are different from element to element; therefore, write the element stiffnesses as K_1 and K_2 for the two elements in the model. We are interested in obtaining the solution to the equilibrium equation in which the externally applied forces, ***P***, are in equilibrium with the internally generated forces, ***I***. When discussing this equation with reference to convergence and nonlinearity, we write it as

$$\{P\} - \{I\} = 0.$$

For the complete two-element, three-node structure we, therefore, modify the signs and rewrite the equilibrium equation as

$$\begin{Bmatrix} P_a \\ P_b \\ P_c \end{Bmatrix} - \begin{bmatrix} K_1 & -K_1 & 0 \\ -K_1 & (K_1+K_2) & -K_2 \\ 0 & -K_2 & K_2 \end{bmatrix} \begin{Bmatrix} u^a \\ u^b \\ u^c \end{Bmatrix} = 0.$$

In an implicit method, such as that used in Abaqus/Standard, this system of equations can then be solved to obtain values for the three unknown variables: u^b, u^c, and P_a (u^a is specified in the problem as 0.0). Once the displacements are known, we can go back and use them to calculate the stresses in the truss elements. Implicit finite element methods require that a system of equations is solved at the end of each solution increment.

In contrast to implicit methods, an explicit method, such as that used in Abaqus/Explicit, does not require the solving of a simultaneous system of equations or the calculation of a global stiffness matrix. Instead, the solution is advanced kinematically from one increment to the next. The extension of the finite element method to explicit dynamics is covered in the following section.

1.6.2 Stress wave propagation illustrated

This section attempts to provide some conceptual understanding of how forces propagate through a model when using the explicit dynamics method. In this illustrative example we consider the propagation of a stress wave along a rod modeled with three elements, as shown in Figure 1–6. We will study the state of the rod as we increment through time.

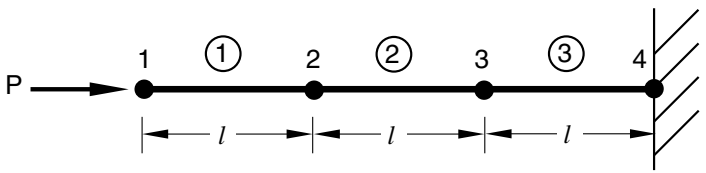
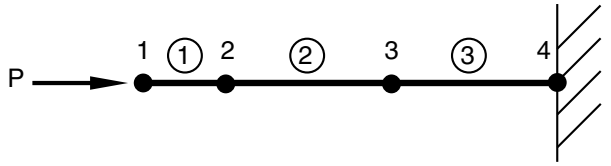


Figure 1–6 Initial configuration of a rod with a concentrated load, P, at the free end.

In the first time increment node 1 has an acceleration, $\ddot{u}_1$, as a result of the concentrated force, P, applied to it. The acceleration causes node 1 to have a velocity, $\dot{u}_1$, which, in turn, causes a strain rate, $\dot{\varepsilon}_{el1}$, in element 1. The increment of strain, $\Delta\varepsilon_{el1}$, in element 1 is obtained by integrating the strain rate through the time of increment 1. The total strain, ε_{el1}, is the sum of the initial strain, ε_0, and the increment in strain. In this case the initial strain is zero. Once the element strain has been calculated, the element stress, σ_{el1}, is obtained by applying the material constitutive model. For a linear elastic material the stress is simply the elastic modulus times the total strain. This process is shown in Figure 1–7. Nodes 2 and 3 do not move in the first increment since no force is applied to them.

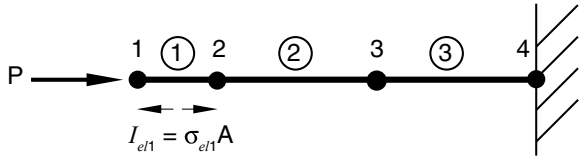


$$\ddot{u}_1 = \frac{P}{M_1} \Rightarrow \dot{u}_1 = \int \ddot{u}_1 dt \Rightarrow \dot{\varepsilon}_{el1} = \frac{-\dot{u}_1}{l} \Rightarrow \Delta\varepsilon_{el1} = \int \dot{\varepsilon}_{el1} dt$$

$$\Rightarrow \varepsilon_{el1} = \varepsilon_o + \Delta\varepsilon_{el1} \Rightarrow \sigma_{el1} = E\varepsilon_{el1}$$

Figure 1–7 Configuration at the end of increment 1 of a rod with a concentrated load, P, at the free end.

In the second increment the stresses in element 1 apply internal, element forces to the nodes associated with element 1, as shown in Figure 1–8. These element stresses are then used to calculate dynamic equilibrium at nodes 1 and 2.



$$\ddot{u}_1 = \frac{P - I_{el1}}{M_1} \Rightarrow \dot{u}_1 = \dot{u}_1^{old} + \int \ddot{u}_1 dt \qquad \dot{\varepsilon}_{el1} = \frac{\dot{u}_2 - \dot{u}_1}{l} \Rightarrow \Delta\varepsilon_{el1} = \int \dot{\varepsilon}_{el1} dt$$

$$\ddot{u}_2 = \frac{I_{el1}}{M_2} \Rightarrow \dot{u}_2 = \int \dot{u}_2 dt \qquad \Rightarrow \varepsilon_{el1} = \varepsilon_{el1}^{old} + \Delta\varepsilon_{el1}$$

$$\Rightarrow \sigma_{el1} = E\varepsilon_{el1}$$

Figure 1–8 Configuration of the rod at the beginning of increment 2.

The process continues so that at the start of the third increment there are stresses in both elements 1 and 2, and there are forces at nodes 1, 2, and 3, as shown in Figure 1–9. The process continues until the analysis reaches the desired total time.

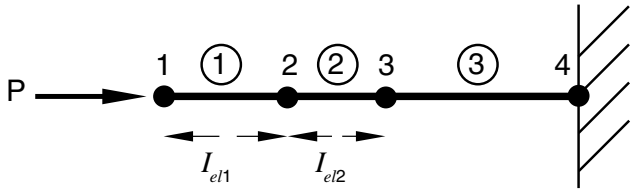
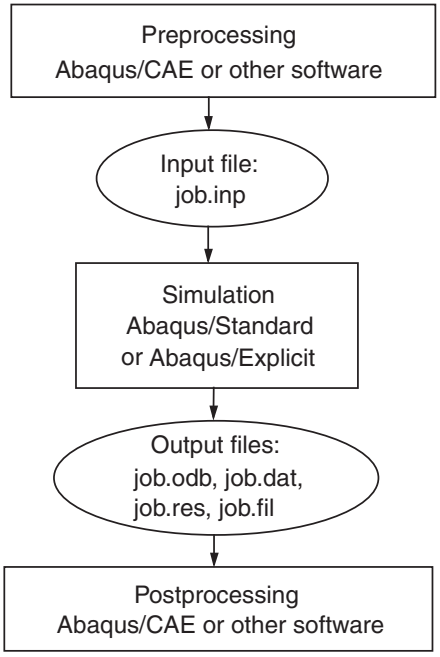


Figure 1–9 Configuration of the rod at the beginning of increment 3.

2. Abaqus Basics

A complete Abaqus analysis usually consists of three distinct stages: preprocessing, simulation, and postprocessing. These three stages are linked together by files as shown below:



Preprocessing (Abaqus/CAE)

In this stage you must define the model of the physical problem and create an Abaqus input file. The model is usually created graphically using Abaqus/CAE or another preprocessor, although the Abaqus input file for a simple analysis can be created directly using a text editor.

Simulation (Abaqus/Standard or Abaqus/Explicit)

The simulation, which normally is run as a background process, is the stage in which Abaqus/Standard or Abaqus/Explicit solves the numerical problem defined in the model. Examples of output from a stress analysis include displacements and stresses that are stored in binary files ready for postprocessing. Depending on the complexity of the problem being analyzed and the power of the computer being used, it may take anywhere from seconds to days to complete an analysis run.

Postprocessing (Abaqus/Viewer)

You can evaluate the results once the simulation has been completed and the displacements, stresses, or other fundamental variables have been calculated. The evaluation is generally done interactively using Abaqus/Viewer or another postprocessor. Abaqus/Viewer, which reads the neutral binary output database file, has a variety of options for displaying the results, including color contour plots, animations, deformed shape plots, and *X–Y* plots.

2.1 Components of an Abaqus analysis model

An Abaqus model is composed of several different components that together describe the physical problem to be analyzed and the results to be obtained. At a minimum the analysis model consists of the following information: discretized geometry, element section properties, material data, loads and boundary conditions, analysis type, and output requests. The discussion in this chapter focuses on structural applications. Similar concepts apply for fluid dynamics.

Discretized geometry

Finite elements and nodes define the basic geometry of the physical structure being modeled in Abaqus. Each element in the model represents a discrete portion of the physical structure, which is, in turn, represented by many interconnected elements. Elements are connected to one another by shared nodes. The coordinates of the nodes and the connectivity of the elements—that is, which nodes belong to which elements—comprise the model geometry. The collection of all the elements and nodes in a model is called the *mesh*. Generally, the mesh will be only an approximation of the actual geometry of the structure.

The element type, shape, and location, as well as the overall number of elements used in the mesh, affect the results obtained from a simulation. The greater the mesh density (i.e., the greater the number of elements in the mesh), the more accurate the results. As the mesh density increases, the analysis results converge to a unique solution, and the computer time required for the analysis increases. The solution obtained from the numerical model is generally an approximation to the solution of the physical problem being simulated. The extent of the approximations made in the model's geometry, material behavior, boundary conditions, and loading determines how well the numerical simulation matches the physical problem.

Element section properties

Abaqus has a wide range of elements, many of which have geometry not defined completely by the coordinates of their nodes. For example, the layers of a composite shell or the dimensions of an I-beam section are not defined by the nodes of the element. Such additional geometric data are defined as physical properties of the element and are necessary to define the model geometry completely (see Chapter 3, "Finite Elements and Rigid Bodies").

Material data

Material properties for all elements must be specified. While high-quality material data are often difficult to obtain, particularly for the more complex material models, the validity of the Abaqus results is limited by the accuracy and extent of the material data.

Loads and boundary conditions

Loads distort the physical structure and, thus, create stress in it. The most common forms of loading include:

- point loads;
- pressure loads on surfaces;
- distributed tractions on surfaces;
- distributed edge loads and moments on shell edges;
- body forces, such as the force of gravity; and
- thermal loads.

Boundary conditions are used to constrain portions of the model to remain fixed (zero displacements) or to move by a prescribed amount (nonzero displacements).

In a static analysis enough boundary conditions must be used to prevent the model from moving as a rigid body in any direction; otherwise, unrestrained rigid body motion causes the stiffness matrix to be singular. A solver problem will occur during the solution stage and may cause the simulation to stop prematurely. Abaqus/Standard will issue a warning message if it detects a solver problem during a simulation. It is important that you learn to interpret such error messages. If you see a "numerical singularity" or "zero pivot" warning message during a static stress analysis, you should check whether all or part of your model lacks constraints against rigid body translations or rotations. Rigid body motions can consist of both translations and rotations of the components. The potential rigid body motions depend on the dimensionality of the model.

Dimensionality	Possible Rigid Body Motion
Three-dimensional	Translation in the 1-, 2-, and 3-directions. Rotation about the 1-, 2-, and 3-axes.
Axisymmetric	Translation in the 2-direction. Rotation about the 3-axis (axisymmetric rigid bodies only).
Plane stress Plane strain	Translation in the 1- and 2-directions. Rotation about the 3-axis.

By default, the 1-, 2-, and 3-directions are aligned with the axes of a global Cartesian coordinate system (discussed later).

In a dynamic analysis inertia forces prevent the model from undergoing infinite motion instantaneously as long as all separate parts in the model have some mass; therefore, solver problem warnings in a dynamic analysis usually indicate some other modeling problem, such as excessive plasticity.

Analysis type

Abaqus can carry out many different types of simulations, but this guide only covers the two most common: static and dynamic stress analyses.

In a static analysis the long-term response of the structure to the applied loads is obtained. In other cases the dynamic response of a structure to the loads may be of interest: for example, the effect of a sudden load on a component, such as occurs during an impact, or the response of a building in an earthquake.

Output requests

An Abaqus simulation can generate a large amount of output. To avoid using excessive disk space, you can limit the output to that required for interpreting the results.

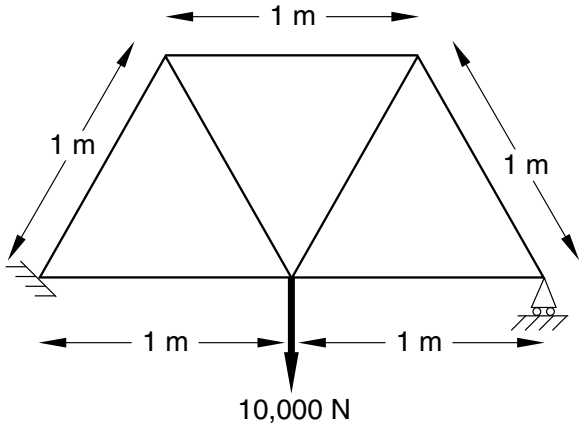
2.2 Format of the input file

The input file is the means of communication between the preprocessor, usually Abaqus/CAE, and the analysis product, Abaqus/Standard or Abaqus/Explicit. It contains a complete description of the numerical model. The input file is a text file that has an intuitive, keyword-based format, so it is easy to modify using a text editor if necessary; if a preprocessor such as Abaqus/CAE is used, modifications should be made using it. Indeed, small analyses can be specified easily by typing the input file directly.

The example of an overhead hoist, shown in Figure 2–1, is used to illustrate the basic format of the Abaqus input file. The hoist is a simple, pin-jointed truss model that is constrained at the left-hand end and mounted on rollers at the right-hand end. The members can rotate freely at the joints. The frame is prevented from moving out of plane. A simulation is performed to determine the structure's deflection and the peak stress in its members when a 10 kN load is applied as shown in Figure 2–1.

Since this problem is very simple, the Abaqus input file is compact and easily understood. The complete Abaqus input file for this example, which is shown in Figure 2–2 and also in "Overhead hoist frame," Section A.1, is split into two distinct parts. The first section contains *model data* and includes all the information required to define the structure being analyzed. The second section contains *history data* that define what happens to the model: the sequence of loading or events for which the response of the structure is required. This history is divided into a sequence of *steps*, each defining a separate part of the simulation. For example, the first step may define a static loading while the second step may define a dynamic loading, etc.

The input file is composed of a number of *option blocks* that contain data describing a part of the model. Each option block begins with a *keyword line*, which is usually followed by one or more *data lines*. These lines cannot exceed 256 characters.



Material properties

General properties:

$\rho = 7800 \text{ kg/m}^3$

Elastic properties:

$E = 200 \times 10^9 \text{ Pa}$

$\nu = 0.3$

Figure 2–1 Schematic of an overhead hoist.

2.2.1 Keyword lines

Keywords (or options) always begin with a star or asterisk (*). For example, *NODE is the keyword for specifying the nodal coordinates, and *ELEMENT is the keyword for specifying the element connectivity. Keywords are often followed by parameters, some of which may be required. The parameter TYPE is required with the *ELEMENT option because the element type must always be given when defining elements. For example, the following statement indicates that we are defining the connectivity of T2D2 elements (two-dimensional truss elements with two nodes):

```
*ELEMENT, TYPE=T2D2
```

Many parameters are optional and are defined only in certain circumstances. For example, the following statement indicates that all the nodes defined in this option block will be put into a set called **PART1**.

```
*NODE, NSET=PART1
```

It is not essential to put nodes into sets, although it is convenient in many instances.

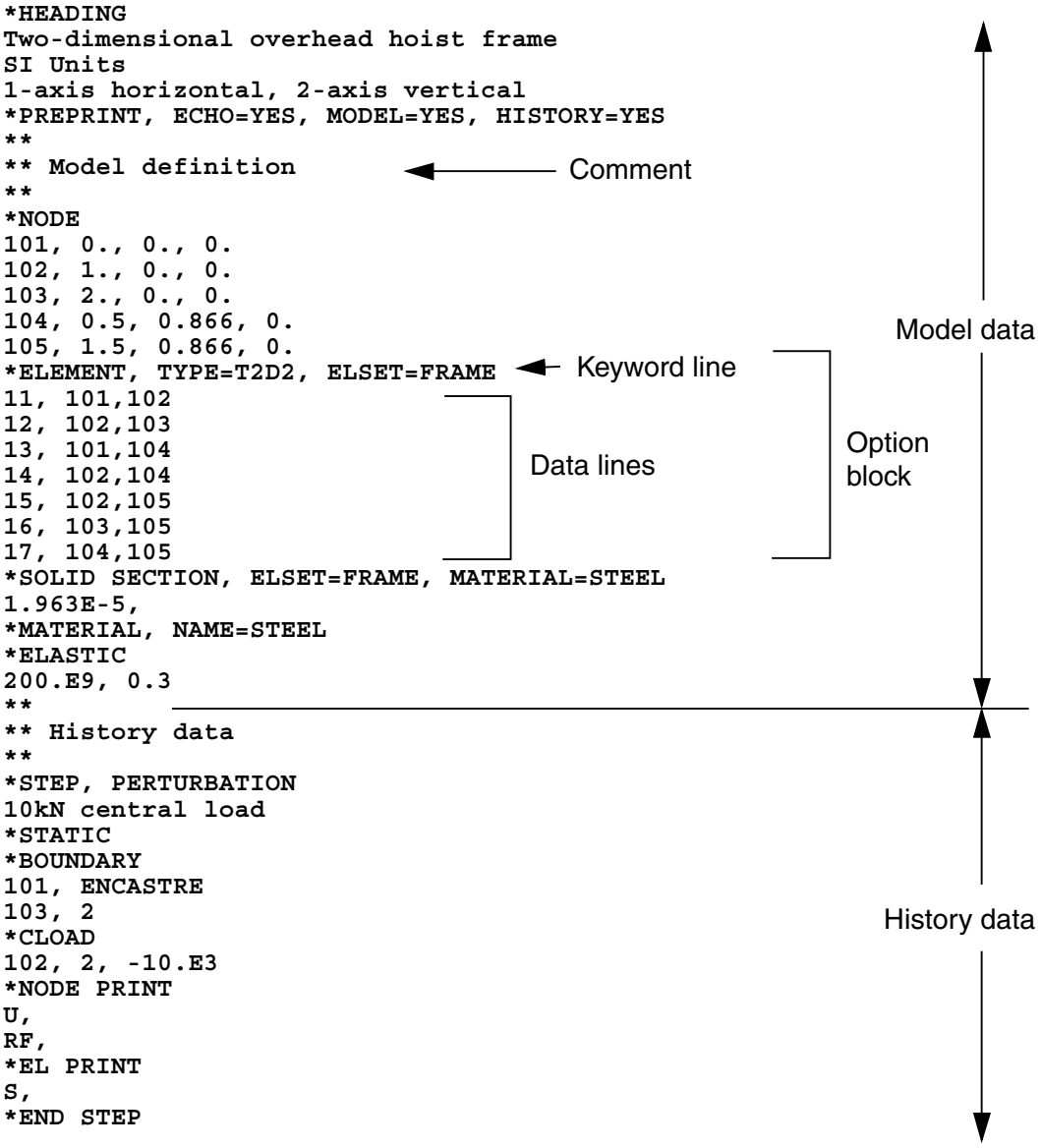


Figure 2–2 Input for overhead hoist model.

Keywords and parameters are case independent and must use enough characters to make them unique. Parameters are separated by commas. If a parameter has a value, an equal sign (=) is used to associate the value with the parameter.

Occasionally, so many parameters are required that they will not fit on a single 256-character line. In this case a comma is placed at the end of the line to indicate that the next line is a continuation line. For example, the following keyword and parameters are a valid keyword line:

```
*ELEMENT, TYPE = T2D2,
ELSET = FRAME
```

Details of the keywords are documented in the Abaqus Keywords Reference Manual.

2.2.2 Data lines

Keyword lines are usually followed by data lines, which provide data that are more easily specified as lists than as parameters on the keyword line. Examples of such data include nodal coordinates; element connectivities; or tables of material properties, such as stress-strain curves. The data required for particular option blocks are specified in the Abaqus Keywords Reference Manual. For example, the option block defining the nodes for the overhead hoist model is:

```
*NODE
101, 0., 0., 0.
102, 1., 0., 0.
103, 2., 0., 0.
104, 0.5, 0.866, 0.
105, 1.5, 0.866, 0.
```

The first piece of data in each data line is an integer that defines the node number. The second, third, and fourth entries are floating-point numbers that specify the x_1, x_2, x_3 coordinates of the node.

The data can consist of a mixture of integer, floating point, or alphanumeric values. Floating point values can be entered in a variety of ways; for example, Abaqus interprets all of the following as the number four:

4.0	4.	4
4.0E+0	.4E+1	40.E−1

Data items are separated by commas, as in Figure 2–2, which allows fairly arbitrary spacing of the input values on the data line. If there is only one item on a data line, it must be followed by a comma.

2.3 Example: creating a model of an overhead hoist

The simulation of the pin-jointed, overhead hoist in Figure 2–1 is used to illustrate the creation of an Abaqus input file using an editor. As you read through this section, you should type the data into a file using one of the editors available on your computer. The Abaqus input file must have an **`.inp`** file extension. For convenience, name the input file **`frame.inp`**. The file identifier, which can be chosen to identify the analysis, is called the *jobname*. In this case use the jobname "frame" to associate it easily with the input file called **`frame.inp`**.

All of the other examples in this guide assume that you will be using a preprocessor, such as Abaqus/CAE, to generate the mesh if you are going to create the model from scratch. Input files for all the examples are available. See Appendix A, "Example Files," for instructions on how to retrieve these input files. However, since the purpose of this example is to help you understand the structure and format of the Abaqus input file, you should type this input file in directly, rather than use a preprocessor or copy the input file that is provided. If you wish to create the entire model using Abaqus/CAE, refer to "Example: creating a model of an overhead hoist," Section 2.3 of Getting Started with Abaqus: Interactive Edition.

2.3.1 Units

Before starting to define this or any model, you need to decide which system of units you will use. Abaqus has no built-in system of units. Do not include unit names or labels when entering data in Abaqus. All input data must be specified in consistent units. Some common systems of consistent units are shown in Table 2–1.

Table 2–1 Consistent units.

Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2/in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	$tonne/mm^3$	$slug/ft^3$	lbf s^2/in^4

The SI system of units is used throughout this guide. Users working in the systems labeled "US Unit" should be careful with the units of density; often the densities given in handbooks of material properties are multiplied by the acceleration due to gravity.

2.3.2 Coordinate systems

You also need to decide which coordinate system to use. The global coordinate system in Abaqus is a right-handed, rectangular (Cartesian) system. For this example define the global 1-axis to be the horizontal axis of the hoist and the global 2-axis to be the vertical axis (Figure 2–3). The global 3-axis is normal to the plane of the framework. The origin (x_1=0, x_2=0, x_3=0) is the bottom left-hand corner of the frame.

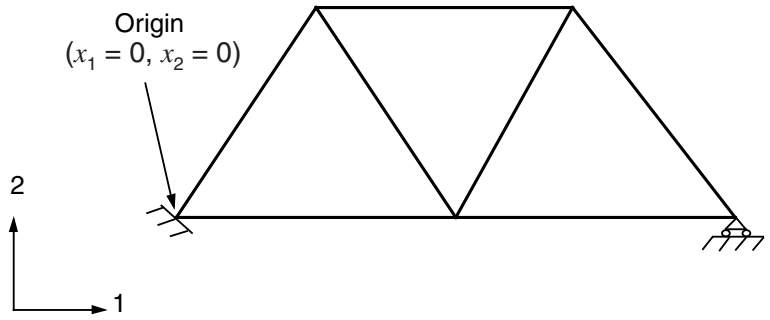


Figure 2–3 Coordinate system and origin for model.

For two-dimensional problems, such as this one, Abaqus requires that the model lie in a plane parallel to the global 1–2 plane.

2.3.3 Mesh

You must select the element types and design the mesh. Creating a proper mesh for a given problem requires experience. For this example you will use a single truss element to model each member of the frame, as shown in Figure 2–4.

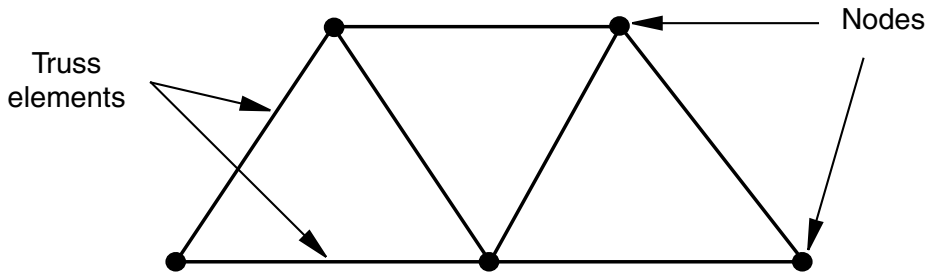


Figure 2–4 Finite element mesh.

A truss element, which can carry only tensile and compressive axial loads, is ideal for modeling pin-jointed frameworks, such as the overhead hoist. Truss elements are described in "Truss elements," Section 3.1.5, and also in the Abaqus Analysis User's Manual, which describes every element available in Abaqus. The index of element types (Section EI.1, "Abaqus/Standard Element Index," of the Abaqus Analysis User's Manual) makes locating a particular element easy. Whenever you are using an element for the first time, you should read the description, which includes the element connectivity and any element section properties needed to define the element's geometry.

The connectivity for the truss elements used in the overhead hoist model is shown in Figure 2–5.

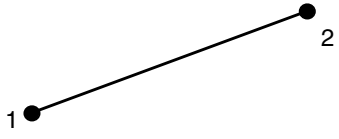


Figure 2–5 Connectivity for the 2-node truss element (T2D2).

Node and element numbers are merely identification labels. They are usually generated automatically by Abaqus/CAE or another preprocessor. The only requirement for node and element numbers is that they must be positive integers. Gaps in the numbering are allowed, and the order in which nodes and elements are defined does not matter. Any nodes that are defined but not associated with an element are removed automatically and are not included in the simulation.

In this case we use the node and element numbers shown in Figure 2–6.

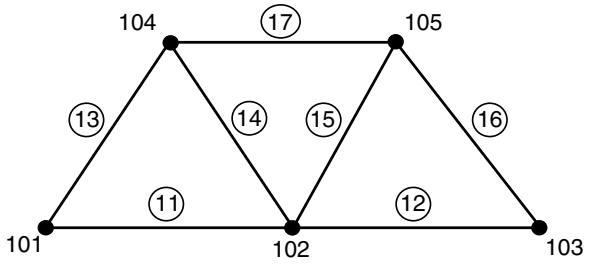


Figure 2–6 Node and element numbers for the hoist model.

2.3.4 Model data

The first part of the input file must contain all of the model data. These data define the structure being analyzed. In the overhead hoist example the model data consist of the following:

- Geometry:
  - Nodal coordinates.
  - Element connectivity.
  - Element section properties.
- Material properties.

Heading

The first option in any Abaqus input file must be *HEADING. The data lines that follow the *HEADING option are lines of text describing the problem being simulated. You should provide an accurate description to allow the input file to be identified at a later date. Moreover, it is often helpful to specify the system of units, directions of the global coordinate system, etc. For example, the *HEADING option block for the hoist problem contains the following:

```
*HEADING
Two-dimensional overhead hoist frame
SI units (kg, m, s, N)
1-axis horizontal, 2-axis vertical
```

Data file printing options

By default, Abaqus will not print an echo of the input file or the model and history definition data to the printed output (**.dat**) file. However, it is recommended that you check your model and history

definition in a **datacheck** run before performing the analysis. The **datacheck** run is discussed later in this chapter.

To request a printout of the input file and of the model and history definition data, add the following statement to the input file:

```
*PREPRINT, ECHO=YES, MODEL=YES, HISTORY=YES
```

Nodal coordinates

The coordinates of each node can be defined once you select the mesh design and node numbering scheme. For this problem use the numbering shown in Figure 2–6. The coordinates of nodes are defined using the ∗NODE option. Each data line of this option block has the form

<node number>,*<*x_1*-coordinate>*,*<*x_2*-coordinate>*,*<*x_3*-coordinate>*

The nodes for the hoist model are defined as follows:

```
*NODE
101, 0.,  0.,    0.
102, 1.,  0.,    0.
103, 2.,  0.,    0.
104, 0.5, 0.866, 0.
105, 1.5, 0.866, 0.
```

Element connectivity

The members of the overhead hoist are modeled with truss elements. The format of each data line for a truss element is

<element number>, *<node 1>*, *<node 2>*

where *node 1* and *node 2* are at the ends of the element (see Figure 2–5). For example, element 16 connects nodes 103 and 105 (see Figure 2–6), so the data line defining this element is

```
16, 103, 105
```

The TYPE parameter on the ∗ELEMENT option must be used to specify the kind of element being defined. In this case you will use T2D2 truss elements.

One of the most useful features in Abaqus is the availability of node and element *sets* that are referred to by name. By using the ELSET parameter on the ∗ELEMENT option, all of the elements defined in the option block are added to an element set called **FRAME**. A set name can have as many as 80 characters and must start with a letter. Since element section properties are assigned through element set names, all elements in the model must belong to at least one element set.

The complete ∗ELEMENT option block for the overhead hoist model (see Figure 2–6) is shown below:

```
*ELEMENT, TYPE=T2D2, ELSET=FRAME
11, 101, 102
```

```
12, 102, 103
13, 101, 104
14, 102, 104
15, 102, 105
16, 103, 105
17, 104, 105
```

Element section properties

Each element must refer to an element section property. The appropriate element section option for each element and the additional geometric data (if any) needed for each element are described in the Abaqus Analysis User's Manual.

For the T2D2 element you must use the *SOLID SECTION option and give one data line with the cross-sectional area of the element. If you leave the data line blank, the cross-sectional area is assumed to be 1.0.

In this case all the members are circular bars that are 5 mm in diameter. Their cross-sectional area is 1.963×10^{-5} m^2.

The MATERIAL parameter, which is required for most element section options, refers to the name of a material property definition that is to be used with the elements. The name can have up to 80 characters and must begin with a letter.

In this example all of the elements have the same section properties and are made of the same material. Typically, there will be several different element section properties in an analysis; for example, different components in a model may be made of different materials. The elements are associated with material properties through element sets. For the overhead hoist model the elements are added to an element set called **FRAME**. Element set **FRAME** is then used as the value of the ELSET parameter on the element section option. Add the following option block to your input file:

```
*SOLID SECTION, ELSET=FRAME, MATERIAL=STEEL
** diameter = 5mm --> area = 1.963E-5 sq. m.
1.963E-5,
```

Any line in the input file that begins with ** is treated as a comment.

Cross-sectional area of truss elements.

Materials

One of the features that makes Abaqus a truly general-purpose finite element program is that almost any material model can be used with any element. Once the mesh has been created, material models can be associated, as appropriate, with the elements in the mesh.

Abaqus has a large number of material models, many of which include nonlinear behavior. In this overhead hoist example we use the simplest form of material behavior: linear elasticity. In Chapter 10, "Materials," two of the most common forms of nonlinear material behavior are considered: metal plasticity and rubber elasticity. A discussion of all the material models available in Abaqus can be found in the Abaqus Analysis User's Manual.

Linear elasticity is appropriate for many materials at small strains, particularly for metals up to their yield point. It is characterized by a linear relationship between stress and strain (Hooke's law), as shown in Figure 2–7.

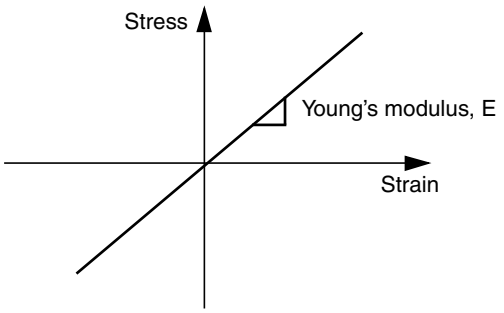


Figure 2–7 Linear elastic material.

The material behavior is characterized by two constants: Young's modulus, E, and Poisson's ratio, ν.

A material definition in the Abaqus input file starts with a *MATERIAL option. The parameter NAME is used to associate a material with an element section property. For example,

```
*SOLID SECTION, ELSET=FRAME, MATERIAL=STEEL
1.963E-5
*MATERIAL, NAME=STEEL
```

Material suboptions directly follow their associated *MATERIAL option. Several suboptions may be required to complete the material definition. All material suboptions are associated with the material that is listed on the most recent *MATERIAL option until another *MATERIAL option or a non-material option block is given.

Without considering thermal expansion effects (which would be defined with the *EXPANSION material suboption), one material suboption, *ELASTIC, is required to define a linear elastic material. The form of this option block is

```
*ELASTIC
<E>,<ν>
```

Therefore, the complete, isotropic, linear elastic material definition for the hoist members, which are made of steel, should be entered into your input file as

```
*MATERIAL, NAME=STEEL
*ELASTIC
200.E9, 0.3
```

The model definition portion of this problem is now complete since all the components describing the structure have been specified.

2.3.5 History data

The history data define the sequence of events for the simulation. This loading history is divided into a series of *steps*, each defining a different portion of the structure's loading. Each step contains the following information:

- the type of simulation (static, dynamic, etc.);
- the loads and constraints; and
- the output required.

In this example we are interested in the static response of the overhead hoist to a 10 kN load applied at the midspan, with the left-hand end fully constrained and a roller constraint on the right-hand end (see Figure 2–1). This is a single event, so only a single step is needed for the simulation.

The ∗STEP option is used to mark the start of a step. Like the ∗HEADING option, this option may be followed by data lines containing a title for the step. In your hoist model use the following ∗STEP option block:

```
*STEP, PERTURBATION
10kN central load
```

The PERTURBATION parameter indicates that this is a linear analysis. If this parameter is omitted, the analysis may be linear or nonlinear. The use of the PERTURBATION parameter is discussed further in Chapter 11, "Multiple Step Analysis."

Analysis procedure

The *analysis procedure* (the type of simulation) must be defined immediately following the ∗STEP option block. In this case we want the long-term static response of the structure. The option for a static simulation is ∗STATIC. For linear analysis this option has no parameters or data lines, so add the following line to your input file:

```
*STATIC
```

The remaining input data in the step define the boundary conditions (constraints), loads, and output required and can be given in any order that is convenient.

Boundary conditions

Boundary conditions are applied to those parts of the model where the displacements are known. Such parts may be constrained to remain fixed (have zero displacement) during the simulation or may have specified, nonzero displacements. In either situation the constraints are applied directly to the nodes of the model.

In some cases a node may be constrained completely and, thus, cannot move in any direction (for example, node 101 in our case). In other cases a node is constrained in some directions but is free to move in others. For example, node 103 is fixed in the vertical direction but is free to move in the horizontal direction. The directions in which a node is able to move are called *degrees of freedom (dof)*. In the case of our two-dimensional hoist, each node can move in the global 1- and 2-directions; therefore, there are two degrees of freedom at each node. If the hoist could move out of plane, the problem would be three-dimensional, and each node would have three degrees of freedom. Nodes attached to beam and shell elements have additional degrees of freedom representing the components of rotation and, thus, may have up to six degrees of freedom.

The labeling convention used for the degrees of freedom in Abaqus is as follows:

1 Translation in the 1-direction (U1).

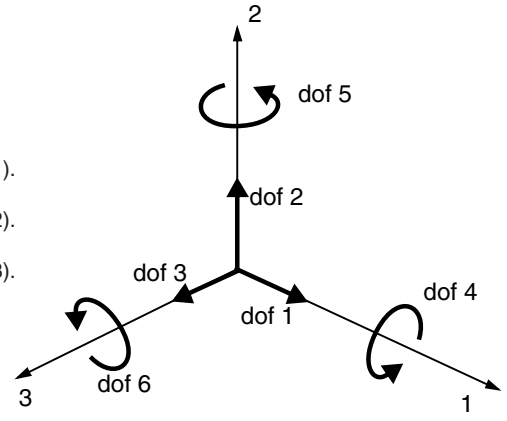
2 Translation in the 2-direction (U2).

3 Translation in the 3-direction (U3).

4 Rotation about the 1-direction (UR1).

5 Rotation about the 2-direction (UR2).

6 Rotation about the 3-direction (UR3).



The degrees of freedom active at a node depend on the type of elements attached to that node. Chapter 3, "Finite Elements and Rigid Bodies," describes the active degrees of freedom for some of the elements available in Abaqus. The two-dimensional truss element, T2D2, has two degrees of freedom active at each node—translation in the 1- and 2-directions (dof 1 and dof 2, respectively).

Constraints on nodes are defined by using the ∗BOUNDARY option and specifying the constrained degrees of freedom. Each data line is of the form:

<node number>, *<first dof>*, *<last dof>*, *<magnitude of displacement>*

The first degree of freedom and last degree of freedom are used to give a range of degrees of freedom that will be constrained. For example, the following statement constrains degrees of freedom 1, 2, and 3 at node 101 to have zero displacement (the node cannot move in either the global 1-, 2-, or 3-direction):

```
101, 1, 3, 0.0
```

If the magnitude of the displacement is not specified on the data line, it is assumed to be zero. If the node is constrained in one direction only, the third field should be blank or equal to the second

field. For example, to constrain node 103 in the 2-direction (degree of freedom 2) only, any of the following data line formats can be used:

```
103, 2,2, 0.0
```

or

```
103, 2,2
```

or

```
103, 2
```

Boundary conditions on a node are cumulative. Thus, the following input constrains node 101 in both directions 1 and 2:

```
101, 1
101, 2
```

Rather than specifying each constrained degree of freedom, some of the more common constraints can be given directly using the following named constraints:

Degree of freedom	Description
ENCASTRE	Constraint on all displacements and rotations at a node.
PINNED	Constraint on all translational degrees of freedom.
XSYMM	Symmetry constraint about a plane of constant x_1.
YSYMM	Symmetry constraint about a plane of constant x_2.
ZSYMM	Symmetry constraint about a plane of constant x_3.
XASYMM	Antisymmetry constraint about a plane of constant x_1.
YASYMM	Antisymmetry constraint about a plane of constant x_2.
ZASYMM	Antisymmetry constraint about a plane of constant x_3.

Thus, another way to constrain all the active degrees of freedom at node 101 in the hoist model is

```
101, ENCASTRE
```

The complete *BOUNDARY option block for our hoist problem is:

```
*BOUNDARY
101, ENCASTRE
103, 2
```

In this example all of the constraints are in the global 1- or 2-directions. In many cases constraints are required in directions that are not aligned with the global directions. The

*TRANSFORM option can be used in such cases to define a local coordinate system for boundary condition application. The skew plate example in Chapter 5, "Using Shell Elements," demonstrates how to use this option in such cases.

Loading

Loading is anything that causes the displacement or deformation of the structure, including:

- concentrated loads,
- pressure loads,
- distributed traction loads,
- distributed edge loads and moment on shells,
- nonzero boundary conditions,
- body loads, and
- temperature (with thermal expansion of the material defined).

In reality there is no such thing as a concentrated, or point, load; the load will always be applied over some finite area. However, if the area being loaded is similar to or smaller than the elements in that area, it is an appropriate idealization to treat the load as a concentrated load applied to a node.

Concentrated loads are specified using the *CLOAD option. The data lines for this option have the form:

<node number>, *<dof>*, *<load magnitude>*

In this simulation a load of −10 kN is applied in the 2-direction to node 102. The option block is:

```
*CLOAD
102, 2, -10.E3
```

Output requests

Finite element analyses can create very large amounts of output. Abaqus allows you to control and manage this output so that only data required to interpret the results of your simulation are produced. Four types of output are available:

- Results stored in a neutral binary file used by Abaqus/Viewer for postprocessing. This file is called the Abaqus output database file and has the extension **.odb**.
- Printed tables of results, written to the Abaqus data (**.dat**) file.
- Restart data, used to continue the analysis, written to the Abaqus restart (**.res**) file.
- Results stored in binary files for subsequent postprocessing with third-party software, written to the Abaqus results (**.fil**) file.

You will use the first two of these in the overhead hoist simulation.

By default, an output database file, which includes a preselected set of the most commonly used output variables for a given type of analysis, is created. A list of preselected variables for default output database output is given in the Abaqus Analysis User's Manual. You do not need

to add any output requests to accept these defaults. For this example the default output database output includes the deformed configuration and the applied nodal loads.

Selected results also can be written in tabular form to the Abaqus data file. By default, no printout is written to the Abaqus data file. The ∗NODE PRINT option controls the printing of nodal results (for example, displacements and reaction forces), while the ∗EL PRINT option controls the printing of element results. A comprehensive list of the output variables available is given in the Abaqus Analysis User's Manual.

The data lines for either of these options list the output to appear in the columns of the table. Each data line creates a separate table of data that can have a maximum of nine columns.

For this analysis we are interested in the displacements of the nodes (output variable U), the reaction forces at the constrained nodes (output variable RF), and the stress in the members (output variable S). Use the following in your input file:

```
*NODE PRINT
U,
RF,
*EL PRINT
S,
```

to request that Abaqus generate three tables of output data in the data file.

Since you have now finished the definition of all the data required for the step, use the ∗END STEP option to mark the end of the step:

```
*END STEP
```

The input file is now complete. Compare the input file you have generated to the complete input file given in Figure 2–2. Save the data as **`frame.inp`**, and exit the editor.

2.3.6 Checking the model

Having generated the input file for this simulation, you are ready to run the analysis. Unfortunately, it is possible to have errors in the input file because of typing errors or incorrect or missing data. You should perform a **datacheck** analysis first before running the simulation. To run a **datacheck** analysis, make sure that you are in the directory where the input file **`frame.inp`** is located, and type the following command:

```
abaqus job=frame datacheck interactive
```

If this command results in an error message, the Abaqus installation on your computer has been customized. You should contact your systems administrator to find out the appropriate command to run Abaqus. The **`job=frame`** parameter specifies that the *jobname* for this analysis is **`frame`**. All the files associated with this analysis will have this *jobname* as their identifier, which allows them to be recognized easily.

The analysis will run interactively, and messages similar to those shown below will appear on your screen:

```
Abaqus JOB frame
Abaqus 6.12-1
Begin Analysis Input File Processor
9/23/2010 9:26:43 AM
Run pre.exe
Abaqus License Manager checked out the following licenses:
Abaqus/Foundation checked out 3 tokens.
9/23/2010 9:26:45 AM
End Analysis Input File Processor
Begin Abaqus/Standard Datacheck
Begin Abaqus/Standard Analysis
9/23/2010 9:26:45 AM
Run standard.exe
Abaqus License Manager checked out the following licenses:
Abaqus/Foundation checked out 3 tokens.
2/23/2010 9:26:45 AM
End Abaqus/Standard Analysis
Abaqus JOB frame COMPLETED
```

When the **datacheck** analysis is complete, you will find that a number of additional files have been created by Abaqus. If any errors are encountered during the **datacheck** analysis, messages will be written to the data file, **`frame.dat`**. This data file is a text file that can be viewed in an editor or printed. Try viewing the data file in a text editor. The file can contain lines up to 256 characters long, so the editor should be able to accommodate that many characters.

Header page

The data file starts with a header page that contains information about the release of Abaqus used to run the analysis. The header page also contains the phone number, address, and contact information of your local office or representative who can offer technical support and advice.

Input file echo

After the header page, the data file includes an echo of the input file. The input data echo is generated by adding the option *PREPRINT, ECHO=YES to the input file. By default, the parameter ECHO is set to NO.

```
                                    A B A Q U S   I N P U T   E C H O

                          5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                     ----------------------------------------------------------------------------------
                     *HEADING
                     Two-dimensional overhead hoist frame
                     SI units (kg, m, s, N)
                     1-axis horizontal, 2-axis vertical
LINE      5          *PREPRINT, ECHO=YES, MODEL=YES, HISTORY=YES
                     **
```

```
                ** Model definition
                **
                *NODE, NSET=NALL
LINE    10      101, 0.,  0.,    0.
                102, 1.,  0.,    0.
                103, 2.,  0.,    0.
                104, 0.5, 0.866, 0.
                105, 1.5, 0.866, 0.
LINE    15      *ELEMENT, TYPE=T2D2, ELSET=FRAME
                11, 101, 102
                12, 102, 103
                13, 101, 104
                14, 102, 104
LINE    20      15, 102, 105
                16, 103, 105
                17, 104, 105
                *SOLID SECTION, ELSET=FRAME, MATERIAL=STEEL
                ** diameter = 5mm --> area = 1.963E-5 m^2
LINE    25      1.963E-5,
                *MATERIAL, NAME=STEEL
                *ELASTIC
                200.E9, 0.3
                **
LINE    30      ** History data
                **
                *STEP, PERTURBATION
                10kN central load
                *STATIC
LINE    35      *BOUNDARY
                101, ENCASTRE
                103, 2
                *CLOAD
                102, 2, -10.E3
LINE    40      *NODE PRINT
                U,
                RF,
                *EL PRINT
                S,
LINE    45      *END STEP
                --------------------------------------------------------------------------------
                    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
                --------------------------------------------------------------------------------
```

Options processed by Abaqus

Following the input data echo is a list of the options processed by Abaqus. This is the first point at which error and warning messages appear. All error messages are prefixed with ***ERROR, while warnings begin with ***WARNING. Since these messages always begin the same way, searching the data file for warning and error messages is straightforward. When the error is a syntax problem (i.e., when Abaqus cannot understand the input), the error message is followed by the line from the input file that is causing the error.

```
OPTIONS BEING PROCESSED
**************************


*HEADING
       Two-dimensional overhead hoist frame
*NODE, NSET=NALL
*ELEMENT, TYPE=T2D2, ELSET=FRAME
*MATERIAL, NAME=STEEL
*ELASTIC
*SOLID SECTION, ELSET=FRAME, MATERIAL=STEEL
*BOUNDARY
*SOLID SECTION, ELSET=FRAME, MATERIAL=STEEL
*STEP, PERTURBATION
*STEP, PERTURBATION
*STEP, PERTURBATION
```

```
10kN central load
*STATIC
*BOUNDARY
*EL PRINT
*EL FILE
*END STEP
*STEP, PERTURBATION
*STATIC
*BOUNDARY
*CLOAD
*NODE PRINT
*NODE FILE
*END STEP
```

Model data

The rest of the data file is a series of tables containing all of the model data and the history data that should be checked for any obvious errors or omissions. These tables are generated by including the option *PREPRINT, MODEL=YES, HISTORY=YES in the input file. However, these tables may take up a large amount of disk space for large models. By default, the parameters MODEL and HISTORY are set to NO.

The model data section begins with the element definitions, which summarize all the model data. The model data also include the material description. It is always a good idea to check that Abaqus has interpreted the material properties you gave in the input file correctly. Mistakes in the material properties can sometimes cause subtle errors that are difficult to detect from the results. It is easier to check the data here.

```
      E L E M E N T   D E F I N I T I O N S

NUMBER    TYPE     PROPERTY    NODES FORMING ELEMENT
                   REFERENCE

  11      T2D2         1        101       102
  12      T2D2         1        102       103
  13      T2D2         1        101       104
  14      T2D2         1        102       104
  15      T2D2         1        102       105
  16      T2D2         1        103       105
  17      T2D2         1        104       105

                              S O L I D   S E C T I O N (S)

PROPERTY NUMBER         1

   MATERIAL NAME                      STEEL
   ATTRIBUTES                          1.96300E-05   0.0000       0.0000

   HOURGLASS CONTROL STIFFNESS    3.84615E+08

   (USED WITH LOWER ORDER REDUCED INTEGRATED SOLID ELEMENTS LIKE CPS4R,CPE4RH,C3D8R)

                        M A T E R I A L   D E S C R I P T I O N

MATERIAL NAME: STEEL

   ELASTIC          YOUNG'S    POISSON'S
                    MODULUS      RATIO
                  2.00000E+11 0.30000
```

```
                                E L E M E N T   S E T S

SET    FRAME
MEMBERS                    11       12       13       14       15       16       17


                                N O D E   S E T S

SET    NALL
MEMBERS                   101      102      103      104      105


                            N O D E   D E F I N I T I O N S

     NODE           COORDINATES                        SINGLE POINT CONSTRAINTS
    NUMBER                                              TYPE    PLUS    DOF

      101     0.0000       0.0000       0.0000       ENCASTRE
      102     1.0000       0.0000       0.0000
      103     2.0000       0.0000       0.0000                    2
      104    0.50000      0.86600       0.0000
      105     1.5000      0.86600       0.0000
```

History data: loads and database output

The history data are presented below in two sections. The first line of the top half of the history data reads 10kN central load, which is the first data line given in the *STEP option block. This line reminds you of the loads applied in this step.

```
     10kN central load

     FIXED TIME INCREMENTS
     TIME INCREMENT IS                                    2.220E-16
     TIME PERIOD IS                                       2.220E-16
GLOBAL STABILIZATION CONTROL IS NOT USED

THIS IS A LINEAR PERTURBATION STEP.
ALL LOADS ARE DEFINED AS CHANGE IN LOAD TO THE REFERENCE STATE

EXTRAPOLATION WILL NOT BE USED

CHARACTERISTIC ELEMENT LENGTH      1.00

DETAILS REGARDING ACTUAL SOLUTION WAVEFRONT REQUESTED

DETAILED OUTPUT OF DIAGNOSTICS TO DATABASE REQUESTED

PRINT OF INCREMENT NUMBER, TIME, ETC., TO THE MESSAGE FILE EVERY     1  INCREMENTS


                          D A T A B A S E   O U T P U T   G R O U P       1

THE FOLLOWING  FIELD   OUTPUT WILL BE WRITTEN EVERY        1 INCREMENT(S)


THE FOLLOWING OUTPUT WILL BE WRITTEN FOR ALL ELEMENTS OF TYPE T2D2.  OUTPUT IS AT THE
INTEGRATION POINTS.

          S         E
```

```
 THE FOLLOWING OUTPUT WILL BE WRITTEN FOR ALL NODES

          U         RF        CF

END OF DATABASE OUTPUT GROUP    1

                          D A T A B A S E   O U T P U T   G R O U P      2

THE FOLLOWING HISTORY  OUTPUT WILL BE WRITTEN EVERY        1 INCREMENT(S)

THE FOLLOWING ENERGY OUTPUT QUANTITIES WILL BE WRITTEN FOR THE WHOLE MODEL

         ALLKE    ALLSE    ALLWK    ALLPD    ALLCD    ALLVD    ALLKL    ALLAE    ALLQB

         ALLEE    ALLIE    ETOTAL   ALLFD    ALLJD    ALLSD    ALLDMD

END OF DATABASE OUTPUT GROUP    2
```

History data: summary

The second half of the history data is displayed below. This section summarizes the element and nodal output requests, boundary conditions, and concentrated loads.

```
                                E L E M E N T   P R I N T

THE FOLLOWING TABLE IS PRINTED AT EVERY 1 INCREMENT FOR ALL ELEMENTS OF TYPE T2D2.  OUTPUT IS AT
THE INTEGRATION POINTS.

  SUMMARIES WILL BE PRINTED WHERE APPLICABLE

       TABLE     1  S11

                             E L E M E N T   F I L E   O U T P U T

 THE FOLLOWING TABLE IS OUTPUT AT EVERY 1 INCREMENT FOR ALL ELEMENTS OF TYPE T2D2.  OUTPUT IS AT
 THE INTEGRATION POINTS.

          S

                                  N O D E   P R I N T

 THE FOLLOWING TABLE IS PRINTED FOR ALL NODES AT EVERY 1 INCREMENT

   SUMMARIES WILL BE PRINTED

        TABLE  1  U1        U2

 THE FOLLOWING TABLE IS PRINTED FOR ALL NODES AT EVERY 1 INCREMENT

   SUMMARIES WILL BE PRINTED

        TABLE  2  RF1       RF2

                            N O D E   F I L E   O U T P U T

THE FOLLOWING TABLE IS OUTPUT FOR ALL NODES AT EVERY 1 INCREMENT

                  U         RF
```

```
                        B O U N D A R Y   C O N D I T I O N S

NODE       DOF      AMP.     MAGNITUDE                NODE       DOF      AMP.     MAGNITUDE
                    REF.                                                   REF.

103          2     (RAMP)      0.0000

- (RAMP) OR (STEP) - INDICATE USE OF DEFAULT AMPLITUDES ASSOCIATED WITH THE STEP

                      B O U N D A R Y   C O N D I T I O N S

NODE    TYPE         NODE     TYPE         NODE     TYPE         NODE     TYPE         NODE     TYPE

101     ENCASTRE

                      C O N C E N T R A T E D   L O A D S

NODE   DOF    AMP.   AMPLITUDE      NODE   DOF    AMP.   AMPLITUDE      NODE   DOF    AMP.   AMPLITUDE
              REF.                                 REF.                                 REF.

102      2            -10000.
```

Remaining items in the data file

If there are any error messages, the number of such messages produced during the **datacheck** analysis is listed at the end of the data file. If there are only warning messages, the number of these messages is listed at the bottom of the data file after any of the requested output.

If error messages are generated during the **datacheck** analysis, it will not be possible to perform the analysis until the causes of the error messages are corrected. The causes of warning messages should always be investigated. Sometimes, warning messages are indications of mistakes in the input data; other times they are harmless and can be ignored safely.

The final section of the data file, not shown in this guide, includes a summary of the size of the numerical model and an estimate of the file sizes required for the simulation. When analyzing large models, use this output to ensure that you have enough disk space available to perform the analysis.

2.3.7 Running the analysis

Make any necessary corrections to your input file. When the **datacheck** analysis completes with no error messages, run the analysis itself by using the command

```
abaqus job=frame continue interactive
```

Messages like those below will appear on the screen:

```
Abaqus JOB frame
Abaqus 6.12-1
Begin Abaqus/Standard Analysis
9/23/2010 9:30:19 AM
```

```
Run standard.exe
Abaqus License Manager checked out the following licenses:
Abaqus/Foundation checked out 3 tokens.
9/23/2010 9:30:20 AM
End Abaqus/Standard Analysis
Abaqus JOB frame COMPLETED
```

You should always perform a **datacheck** analysis before running a simulation to ensure that the input data are correct and to check that there is enough disk space and memory available to complete the analysis. However, it is possible to combine the **datacheck** and analysis phases of the simulation by using the command

```
abaqus job=frame interactive
```

If a simulation is expected to take a substantial amount of time, it is convenient to run it in the background by omitting the **interactive** parameter:

```
abaqus job=frame
```

(The above commands apply for the standard Abaqus installation on a workstation. However, Abaqus jobs may be run in batch queues on some computers. If you have any questions, ask your systems administrator how to run Abaqus on your system.)

2.3.8 Results

After the analysis is completed, the data file, **frame.dat**, will contain the tables of results requested with the *NODE PRINT and *EL PRINT options. The tables of results follow the output from the **datacheck** analysis. The results from the overhead hoist simulation follow.

Element output

```
Two-dimensional overhead hoist frame                              STEP    1  INCREMENT    1
10kN central load                                    TIME COMPLETED IN THIS STEP   0.00


                    S T E P      1    S T A T I C   A N A L Y S I S


          10kN central load

          FIXED TIME INCREMENTS
          TIME INCREMENT IS                                  2.220E-16
          TIME PERIOD IS                                     2.220E-16

     LINEAR EQUATION SOLVER TYPE         DIRECT SPARSE

     THIS IS A LINEAR PERTURBATION STEP.
     ALL LOADS ARE DEFINED AS CHANGE IN LOAD TO THE REFERENCE STATE
```

```
                    M E M O R Y   E S T I M A T E

PROCESS      FLOATING PT        MINIMUM MEMORY        MEMORY TO
             OPERATIONS           REQUIRED          MINIMIZE I/O
           PER ITERATION          (MBYTES)           (MBYTES)

    1         2.65E+002               13                 20
NOTE:
     (1) THE ESTIMATE PRINTED IS THE MAXIMUM ESTIMATE FROM THE CURRENT STEP TO THE LAST STEP
         OF THE ANALYSIS, WITH THE UNSYMMETRIC MATRIX AND SOLVER TAKEN INTO ACCOUNT IF
         APPLICABLE. SINCE THE ESTIMATE IS BASED ON THE ACTIVE DEGREES OF FREEDOM IN THE
         FIRST ITERATION OF THE CURRENT STEP, FOR PROBLEMS WITH SUBSTANTIAL CHANGES IN ACTIVE
         DEGREES OF FREEDOM BETWEEN STEPS (OR EVEN WITHIN THE SAME STEP), THE MEMORY ESTIMATE
         MIGHT BE NOTICEABLY DIFFERENT THAN THE ACTUAL USAGE. A FEW EXAMPLES ARE: PROBLEMS
         WITH SIGNIFICANT CONTACT CHANGES, PROBLEMS WITH MODEL CHANGE, PROBLEMS WITH BOTH
         STATIC STEP AND STEADY STATE DYNAMIC PROCEDURES, WHERE THE ACOUSTIC ELEMENTS WILL
         ONLY BE ACTIVATED IN THE STEADY STATE DYNAMIC STEPS.
     (2) THE ESTIMATE FOR THE FLOATING POINT OPERATIONS ON EACH PROCESS IS BASED
         ON THE INITIAL LOAD SCHEDULING AND THIS MIGHT NOT REFLECT THE ACTUAL FLOATING
         POINT OPERATIONS COMPLETED ON EACH PROCESS. DUE TO THE DYNAMIC LOAD BALANCING SCHEME,
         THE ACTUAL LOAD BALANCE IS EXPECTED TO BE BETTER THAN THE ESTIMATE PRINTED HERE.
     (3) DEPENDING ON THE SETTING OF THE memory PARAMETER, THE DISK USAGE BY SCRATCH DATA CAN
         VARY FROM CLOSE TO ZERO TO THE ESTIMATED MEMORY TO MINIMIZE I/O.
     (4) USING RESTART, WRITE CAN GENERATE A LARGE AMOUNT OF DATA.

                                  INCREMENT     1 SUMMARY

 TIME INCREMENT COMPLETED  2.220E-16,  FRACTION OF STEP COMPLETED   1.00
 STEP TIME COMPLETED       2.220E-16,  TOTAL TIME COMPLETED         0.00

                              E L E M E N T   O U T P U T

THE FOLLOWING TABLE IS PRINTED FOR ALL ELEMENTS WITH TYPE T2D2 AT THE INTEGRATION POINTS

    ELEMENT  PT FOOT-       S11
                NOTE

          11   1         1.4706E+08
          12   1         1.4706E+08
          13   1        -2.9412E+08
          14   1         2.9412E+08
          15   1         2.9412E+08
          16   1        -2.9412E+08
          17   1        -2.9412E+08

 MAXIMUM                2.9412E+08
 ELEMENT                    14

 MINIMUM               -2.9412E+08
 ELEMENT                    17
```

Node output

```
               N O D E   O U T P U T

   THE FOLLOWING TABLE IS PRINTED FOR ALL NODES

       NODE FOOT-    U1          U2
            NOTE

        102      7.3531E-04 -4.6698E-03
        103      1.4706E-03   0.000
        104      1.4706E-03 -2.5472E-03
        105       0.000     -2.5472E-03
```

```
MAXIMUM          1.4706E-03   0.000
AT NODE               104          101

MINIMUM           0.000      -4.6698E-03
AT NODE               101          102


  THE FOLLOWING TABLE IS PRINTED FOR ALL NODES

      NODE FOOT-    RF1          RF2
           NOTE

       101      -9.0949E-13   5000.
       103        0.000       5000.

MAXIMUM            0.000       5000.
AT NODE               102          103

MINIMUM          -9.0949E-13   0.000
AT NODE               101          102
```

Are the nodal displacements and peak stresses in the individual members reasonable for this hoist and these applied loads?

It is always a good idea to check that the results of the simulation satisfy basic physical principles. In this case check that the external forces applied to the hoist sum to zero in both the vertical and horizontal directions.

What nodes have vertical forces applied to them? What nodes have horizontal forces? Do the results from your simulation match those shown here?

Abaqus also creates several other files during a simulation. One such file—the output database file, **`frame.odb`**—can be used to visualize the results graphically using Abaqus/Viewer.

2.3.9 Postprocessing

Graphical postprocessing is important because of the great volume of data created during a simulation. For any realistic model it is impractical for you to try to interpret results in the tabular form of the data file. Abaqus/Viewer allows you to view the results graphically using a variety of methods, including deformed shape plots, contour plots, vector plots, animations, and *X*–*Y* plots. All of these methods are discussed in this guide. For more information on any of the postprocessing features discussed in this guide, consult the sections on the Visualization module in the Abaqus/CAE User's Manual. For this example you will use Abaqus/Viewer to do some basic model checks and to display the deformed shape of the frame.

Start Abaqus/Viewer by typing the following command at the operating system prompt:

```
abaqus viewer
```

The Abaqus/Viewer window appears.

To begin this exercise, open the output database file that Abaqus/Standard generated during the analysis of the problem.

To open the output database file:

1. From the main menu bar, select **File→Open**; or use the tool in the **File** toolbar.
   The **Open Database** dialog box appears.
2. From the list of available output database files, select **`frame.odb`**.
3. Click **OK**.

   Tip: You can also open the output database **`frame.odb`** by typing the following command at the operating system prompt:

   ```
   abaqus viewer odb=frame
   ```

   Abaqus/Viewer opens the output database created by the job and displays the undeformed model shape, as shown in Figure 2–8.

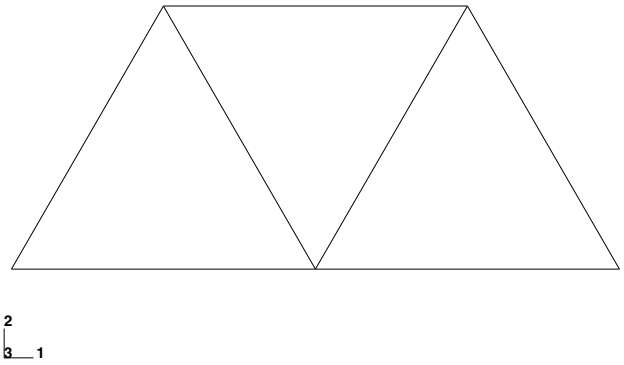


Figure 2–8 Undeformed model shape.

You can choose to display the title block and state block at the bottom of the viewport; these blocks are not shown in Figure 2–8. The title block at the bottom of the viewport indicates the following:

- The description of the model (from the job description).
- The name of the output database (from the name of the analysis job).
- The product name (Abaqus/Standard or Abaqus/Explicit) and release used to generate the output database.
- The date the output database was last modified.

The state block at the bottom of the viewport indicates the following:

- Which step is being displayed.
- The increment within the step.
- The step time.

The view orientation triad indicates the orientation of the model in the global coordinate system. The 3D compass located in the upper-right corner of the viewport allows you to manipulate the view directly.

You can suppress the display of and customize the title block, state block, view orientation triad, and 3D compass by selecting **Viewport→Viewport Annotation Options** from the main menu bar (for example, many of the figures in this manual do not include the title block or the compass).

The Results Tree

You will use the Results Tree to query the components of the model. The Results Tree allows easy access to the history output contained in an output database file for the purpose of creating *X–Y* plots and also to groups of elements, nodes, and surfaces based on set names, material and section assignment, etc. for the purposes of verifying the model and also controlling the viewport display.

To query the model:

1. All output database files that are open in a given postprocessing session are listed underneath the **Output Databases** container. Expand this container and then expand the container for the output database named **`frame.odb`**.
2. Expand the **Materials** container, and click the material named **STEEL**.

   All elements are highlighted in the viewport because only one material assignment was used in this analysis.

The Results Tree will be used more extensively in later examples to illustrate the *X–Y* plotting capability and manipulating the display using display groups.

Customizing an undeformed shape plot

You will now use the plot options to enable the display of node and element numbering. Plot options that are common to all plot types (undeformed, deformed, contour, symbol, and material orientation) are set in a single dialog box. The contour, symbol, and material orientation plot types have additional options, each specific to the given plot type.

To display node numbers:

1. From the main menu bar, select **Options→Common**; or use the tool in the toolbox.

   The **Common Plot Options** dialog box appears.
2. Click the **Labels** tab.
3. Toggle on **Show node labels**.
4. Click **Apply**.

   Abaqus/Viewer applies the change and keeps the dialog box open.

   The customized undeformed plot is shown in Figure 2–9.

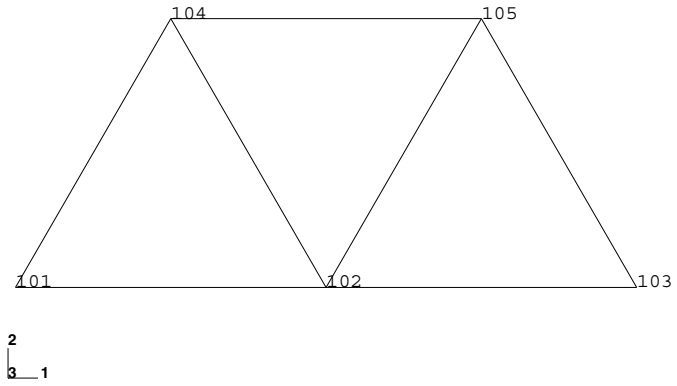


Figure 2–9 Node number plot.

To display element numbers:

1. In the **Labels** tabbed page of the **Common Plot Options** dialog box, toggle on **Show element labels**.
2. Click **OK**.

   Abaqus/Viewer applies the change and closes the dialog box.

   The resulting plot is shown in Figure 2–10.

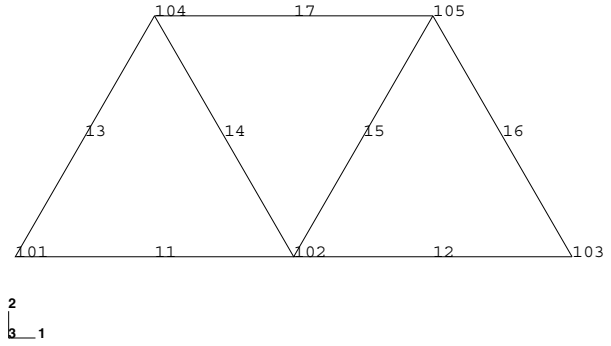


Figure 2–10 Node and element number plot.

Remove the node and element labels before proceeding. To disable the display of node and element numbers, repeat the above procedure and, under **Labels**, toggle off **Show node labels** and **Show element labels**.

Displaying and customizing a deformed shape plot

You will now display the deformed model shape and use the plot options to change the deformation scale factor. You will also superimpose the undeformed model shape on the deformed model shape.

From the main menu bar, select **Plot→Deformed Shape**; or use the tool in the toolbox. Abaqus/Viewer displays the deformed model shape, as shown in Figure 2–11.

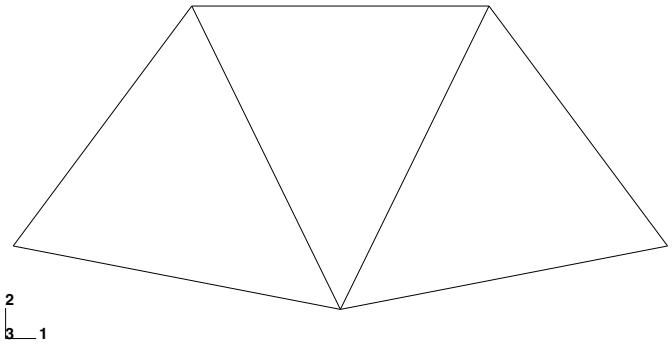


Figure 2–11 Deformed model shape.

For small-displacement analyses (the default formulation in Abaqus/Standard) the displacements are scaled automatically to ensure that they are clearly visible. The scale factor is displayed in the state block. In this case the displacements have been scaled by a factor of 42.83.

To change the deformation scale factor:

1. From the main menu bar, select **Options→Common**; or use the tool in the toolbox.
2. From the **Common Plot Options** dialog box, click the **Basic** tab if it is not already selected.
3. From the **Deformation Scale Factor** area, toggle on **Uniform** and enter `10.0` in the **Value** field.
4. Click **Apply** to redisplay the deformed shape.

   The state block displays the new scale factor.
5. To return to automatic scaling of the displacements, repeat the above procedure and, in the **Deformation Scale Factor** field, toggle on **Auto-compute**.
6. Click **OK** to close the **Common Plot Options** dialog box.

To superimpose the undeformed model shape on the deformed model shape:

1. Click the **Allow Multiple Plot States** tool in the toolbox to allow multiple plot states in the viewport; then click the tool or select **Plot→Undeformed Shape** to add the undeformed shape plot to the existing deformed plot in the viewport.

   By default, Abaqus/Viewer plots the deformed model shape in green and the (superimposed) undeformed model shape in a translucent white.
2. The plot options for the superimposed image are controlled separately from those of the primary image. From the main menu bar, select **Options→Superimpose**; or use the tool in the toolbox to change the edge style of the superimposed (i.e., undeformed) image.
3. From the **Superimpose Plot Options** dialog box, click the **Color & Style** tab.
4. In the **Color & Style** tabbed page, select the dashed edge style.
5. Click **OK** to close the **Superimpose Plot Options** dialog box and to apply the change.

The plot is shown in Figure 2–12. The undeformed model shape appears with a dashed edge style.

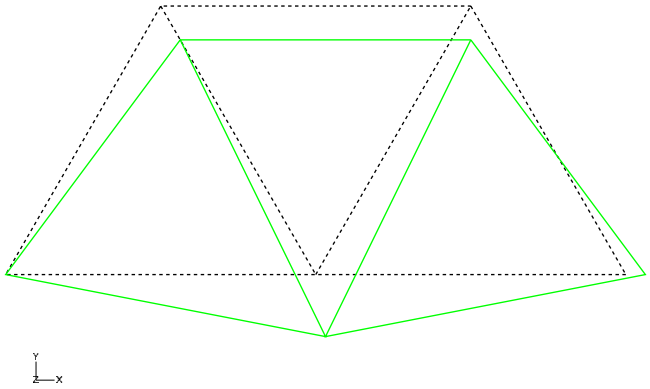


Figure 2–12 Undeformed and deformed model shapes.

Checking the model with Abaqus/Viewer

You can use Abaqus/Viewer to check that the model is correct before running the simulation. You have already learned how to draw plots of the model and to display the node and element numbers. These are useful tools for checking that Abaqus is using the correct mesh.

The boundary conditions applied to the overhead hoist model can also be displayed and checked.

To display boundary conditions on the undeformed model:

1. Click the tool in the toolbox to disable multiple plot states in the viewport.
2. Display the undeformed model shape, if it is not displayed already.
3. From the main menu bar, select **View→ODB Display Options**.
4. In the **ODB Display Options** dialog box, click the **Entity Display** tab.
5. Toggle on **Show boundary conditions**.
6. Click **OK**.

   Abaqus/Viewer displays symbols to indicate the applied boundary conditions, as shown in Figure 2–13.

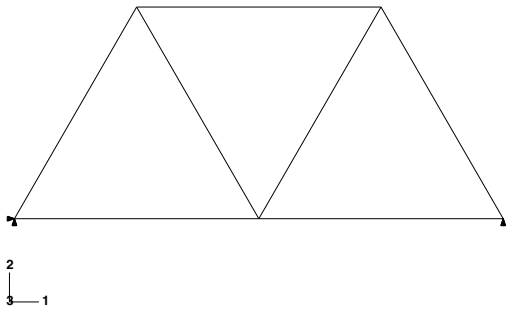


Figure 2–13 Applied boundary conditions on the overhead hoist.

Tabular data reports

In addition to the graphical capabilities described above, Abaqus/Viewer allows you to write data to a text file in a tabular format. This is a convenient alternative to writing printed data to the data (`.dat`) file, especially for complicated models. Output generated this way has many uses; for example, it can be used in written reports. In this problem you will generate a report containing the element stresses, nodal displacements, and reaction forces.

To generate field data reports:

1. From the main menu bar, select **Report→Field Output**.

2. In the **Variable** tabbed page of the **Report Field Output** dialog box, accept the default position labeled **Integration Point**. Click the triangle next to **S: Stress components** to expand the list of available variables. From this list, toggle on **S11**.
3. In the **Setup** tabbed page, name the report **`Frame.rpt`**. In the **Data** region at the bottom of the page, toggle off **Column totals**.
4. Click **Apply**.

   The element stresses are written to the report file.
5. In the **Variable** tabbed page of the **Report Field Output** dialog box, change the position to **Unique Nodal**. Toggle off **S: Stress components**, and select **U1** and **U2** from the list of available **U: Spatial displacement** variables.
6. Click **Apply**.

   The nodal displacements are appended to the report file.
7. In the **Variable** tabbed page of the **Report Field Output** dialog box, toggle off **U: Spatial displacement**, and select **RF1** and **RF2** from the list of available **RF: Reaction force** variables.
8. In the **Data** region at the bottom of the **Setup** tabbed page, toggle on **Column totals**.
9. Click **OK**.

   The reaction forces are appended to the report file, and the **Report Field Output** dialog box closes.

Open the file **`Frame.rpt`** in a text editor. The contents of this file are shown below. Your node and element numbering may be different. Very small values may also be calculated differently, depending on your system.

Stress output:

```
Field Output Report

Source 1
---------

   ODB: frame.odb
   Step: Step-1
   Frame: Increment      1: Step Time =   2.2200E-16

Loc 1 : Integration point values from source 1

Output sorted by column "Element Label".

Field Output reported at integration points for part: PART-1-1

         Element             Int           S.S11
           Label              Pt          @Loc 1
-------------------------------------------------
              11               1     147.062E+06
              12               1     147.062E+06
              13               1    -294.118E+06
              14               1     294.118E+06
              15               1     294.118E+06
              16               1    -294.118E+06
              17               1    -294.125E+06
```

```
  Minimum                                -294.125E+06
      At Element                                 17

          Int Pt                                  1
  Maximum                                 294.118E+06
      At Element                                 15

          Int Pt                                  1
```

Displacement output:

```
Field Output Report

Source 1
---------

   ODB: frame.odb
   Step: Step-1
   Frame: Increment      1: Step Time =   2.2200E-16

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: PART-1-1

            Node            U.U1            U.U2
           Label         @Loc 1          @Loc 1
-------------------------------------------------
             101              0.         -5.E-33
             102     735.312E-06    -4.66977E-03
             103     1.47062E-03         -5.E-33
             104     1.47062E-03    -2.54716E-03
             105     433.681E-21    -2.54716E-03


  Minimum                    0.    -4.66977E-03

         At Node             101             102
  Maximum            1.47062E-03         -5.E-33

         At Node             104             103
```

Reaction force output:

```
Field Output Report

Source 1
---------

   ODB: frame.odb
   Step: Step-1
   Frame: Increment      1: Step Time =   2.2200E-16

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: PART-1-1

            Node          RF.RF1          RF.RF2
           Label         @Loc 1          @Loc 1
-------------------------------------------------
             101    -909.495E-15         5.E+03
             102              0.             0.
             103              0.         5.E+03
             104              0.             0.
```

```
           105              0.              0.


Minimum           -909.495E-15              0.
       At Node             101             105

Maximum                     0.          5.E+03
       At Node             105             103

         Total    -909.495E-15         10.E+03
```

The information obtained in these tables is the same as that examined earlier when reviewing the printed results in the data (**.dat**) file. The advantage of using Abaqus/Viewer to generate the tabular data is that you may create it as a postprocessing operation, whereas writing it to the data (**.dat**) file requires you to include the appropriate option in the input file (a preprocessing operation). Thus, Abaqus/Viewer offers greater flexibility to generate tabular output.

2.3.10 Rerunning the analysis using Abaqus/Explicit

We will rerun the same analysis in Abaqus/Explicit for comparison. This time we are interested in the dynamic response of the hoist to the same load applied suddenly at the midspan. Before continuing, save a copy of **frame.inp** as **frame_xpl.inp**. Make all subsequent changes to the **frame_xpl.inp** input file. You will need to replace the static step with an explicit dynamic step, modify the output requests and the material definition, and change the element library before you can resubmit the job.

Modifying the material definition

Since Abaqus/Explicit performs a dynamic analysis, a complete material definition requires that you specify the material density. For this problem assume the density is equal to 7800 kg/m^3.

You can modify the material definition by adding the *DENSITY option to the material option block. The form of this option is as follows:

```
*DENSITY
<ρ>,
```

Thus, the complete material definition for the hoist members is:

```
*MATERIAL, NAME=STEEL
*ELASTIC
200.E9, 0.3
*DENSITY
7800.,
```

Replacing the analysis step

The step definition must change to reflect a dynamic, explicit analysis. Locate the existing *STEP option block, which appears as follows:

```
*STEP, PERTURBATION
10kN central load
```

Replace this option block with the following one:

```
*STEP
10kN central load, suddenly applied
```

The *analysis procedure* (the type of simulation) must be defined immediately following the *STEP option block. In Abaqus/Explicit the three analysis options are *DYNAMIC, EXPLICIT; *DYNAMIC TEMPERATURE-DISPLACEMENT, EXPLICIT; and *ANNEAL. The *DYNAMIC TEMPERATURE-DISPLACEMENT procedure simulates the fully coupled thermal-mechanical response of a body, while the *ANNEAL procedure simulates the relaxation of stresses and plastic strains that occurs as metals are heated to a high temperature. In this simulation we want to determine the dynamic response of the structure over a period of 0.01 s. Thus, we will use *DYNAMIC, EXPLICIT. Replace the *STATIC option block with the following:

```
*DYNAMIC, EXPLICIT
, 0.01
```

Modifying the output requests

Because this is a dynamic analysis in which the transient response of the frame is of interest, it is helpful to have the displacements of the center point written as history output. Displacement history output can be requested only for a node set. Thus, you will create a node set that contains the node at the center of the bottom of the truss. Then you will add displacements to the history output requests.

Create a set named **CENTER** using the *NSET option, as follows:

```
*NSET, NSET=CENTER
102,
```

Place this option block in the model data portion of your input file (e.g., after the node definitions).

Replace the existing output requests with the following:

```
*OUTPUT, FIELD, VARIABLE=PRESELECT
*OUTPUT, HISTORY, VARIABLE=PRESELECT, FREQUENCY=1
*NODE OUTPUT, NSET=CENTER
U,
```

Submitting the new input file for analysis

Perform an interactive **datacheck** analysis of the input data in the **frame_xpl** input file:

```
abaqus job=frame_xpl datacheck interactive
```

Make any necessary corrections to your input file. When the **datacheck** analysis completes with no error messages, run the analysis itself by using the command

```
abaqus job=frame_xpl continue interactive
```

2.3.11 Postprocessing the dynamic analysis results

For the static linear perturbation analysis done in Abaqus/Standard you examined the deformed shape as well as stress, displacement, and reaction force output. For the Abaqus/Explicit analysis you can similarly examine the deformed shape and generate field data reports. Because this is a dynamic analysis, you should also examine the transient response resulting from the loading. You will do this by animating the time history of the deformed model shape and plotting the displacement history of the bottom center node in the truss.

Start by opening the **`frame_xpl`** output database using the instructions in "Postprocessing," Section 2.3.9, then plot the deformed shape of the model. For large-displacement analyses (the default formulation in Abaqus/Explicit) the displaced shape scale factor has a default value of 1. Change the **Deformation Scale Factor** to 20 so that you can more easily see the deformation of the truss.

To create a time-history animation of the deformed model shape:

1. From the main menu bar, select **Animate→Time History**; or use the tool in the toolbox.
   The time history animation begins in a continuous loop at its fastest speed. Abaqus/Viewer displays the movie player controls in the right side of the context bar (immediately above the viewport).
2. From the main menu bar, select **Options→Animation**; or use the animation options tool in the toolbox (located directly underneath the tool).
   The **Animation Options** dialog box appears.
3. Change the **Mode** to **Play Once**, and slow the animation down by moving the **Frame Rate** slider.
4. You can use the animation controls to start, pause, and step through the animation. From left to right of Figure 2–14, these controls perform the following functions: **play/pause**, **first**, **previous**, **next**, and **last**.

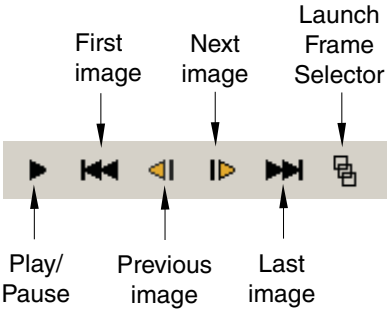


Figure 2–14 Postprocessing animation controls.

The truss responds dynamically to the load. You can confirm this by plotting the vertical displacement history of the node set **CENTER**.

You can create *X–Y* curves from either history or field data stored in the output database (**.odb**) file. *X–Y* curves can also be read from an external file or they can be typed into Abaqus/Viewer interactively. Once curves have been created, their data can be further manipulated and plotted to the screen in graphical form. In this example you will create and plot the curve using history data.

To create an *X–Y* plot of the vertical displacement for a node:

1. In the Results Tree, expand the **History Output** container underneath the output database named **frame_xpl.odb**.
2. From the list of available history output, double-click **Spatial displacement: U2 at Node 102 in NSET CENTER**.

   Abaqus/Viewer plots the vertical displacement at the center node along the bottom of the truss, as shown in Figure 2–15.

   Note: The chart legend has been suppressed and the axis labels modified in this figure. Many *X–Y* plot options are directly accessible by double-clicking the appropriate regions of the viewport. To enable direct object actions, however, you must first click in the prompt area to cancel the current procedure (if necessary). To suppress the legend, double-click it in the viewport to open the **Chart Legend Options** dialog box. In the **Contents** tabbed page of this dialog box, toggle off **Show legend**. To modify the axis labels, double-click either axis to open the **Axis Options** dialog box, and edit the axis titles as indicated in Figure 2–15.

Exiting Abaqus/Viewer

Save your model database file; then select **File→Exit** from the main menu bar to exit Abaqus/Viewer.

2.4 Comparison of implicit and explicit procedures

Abaqus/Standard and Abaqus/Explicit are capable of solving a wide variety of problems. The characteristics of implicit and explicit procedures determine which method is appropriate for a given problem. For those problems that can be solved with either method, the efficiency with which the problem can be solved can determine which product to use. Understanding the characteristics of implicit and explicit procedures will help you answer this question. Table 2–2 lists the key differences between the analysis products, which are discussed in detail in the relevant chapters in this guide.

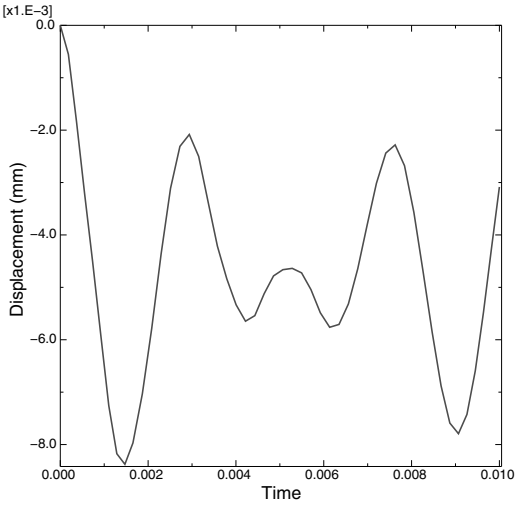


Figure 2–15 Vertical displacement at the midspan of the truss.

Table 2–2 Key differences between Abaqus/Standard and Abaqus/Explicit.

Quantity	Abaqus/Standard	Abaqus/Explicit
Element library	Offers an extensive element library.	Offers an extensive library of elements well suited for explicit analyses. The elements available are a subset of those available in Abaqus/Standard.
Analysis procedures	General and linear perturbation procedures are available.	General procedures are available.
Material models	Offers a wide range of material models.	Similar to those available in Abaqus/Standard; a notable difference is that failure material models are allowed.
Contact formulation	Has a robust capability for solving contact problems.	Has a robust contact functionality that readily solves even the most complex contact simulations.

Quantity	Abaqus/Standard	Abaqus/Explicit
Solution technique	Uses a stiffness-based solution technique that is unconditionally stable.	Uses an explicit integration solution technique that is conditionally stable.
Disk space and memory	Due to the large numbers of iterations possible in an increment, disk space and memory usage can be large.	Disk space and memory usage is typically much smaller than that for Abaqus/Standard.

2.4.1 Choosing between implicit and explicit analysis

For many analyses it is clear whether Abaqus/Standard or Abaqus/Explicit should be used. For example, as demonstrated in Chapter 8, "Nonlinearity," Abaqus/Standard is more efficient for solving smooth nonlinear problems; on the other hand, Abaqus/Explicit is the clear choice for a wave propagation analysis. There are, however, certain static or quasi-static problems that can be simulated well with either program. Typically, these are problems that usually would be solved with Abaqus/Standard but may have difficulty converging because of contact or material complexities, resulting in a large number of iterations. Such analyses are expensive in Abaqus/Standard because each iteration requires a large set of linear equations to be solved.

Whereas Abaqus/Standard must iterate to determine the solution to a nonlinear problem, Abaqus/Explicit determines the solution without iterating by *explicitly* advancing the kinematic state from the end of the previous increment. Even though a given analysis may require a large number of time increments using the explicit method, the analysis can be more efficient in Abaqus/Explicit if the same analysis in Abaqus/Standard requires many iterations.

Another advantage of Abaqus/Explicit is that it requires much less disk space and memory than Abaqus/Standard for the same simulation. For problems in which the computational cost of the two programs may be comparable, the substantial disk space and memory savings of Abaqus/Explicit make it attractive.

2.4.2 Cost of mesh refinement in implicit and explicit analyses

Using the explicit method, the computational cost is proportional to the number of elements and roughly inversely proportional to the smallest element dimension. Mesh refinement, therefore, increases the computational cost by increasing the number of elements and reducing the smallest element dimension. As an example, consider a three-dimensional model with uniform, square elements. If the mesh is refined by a factor of two in all three directions, the computational cost increases by a factor of $2 \times 2 \times 2$ as a result of the increase in the number of elements and by a factor of 2 as a result of the decrease in the smallest element dimension. The total computational cost of the analysis increases by a factor of 2^4,

or 16, by refining the mesh. Disk space and memory requirements are proportional to the number of elements with no dependence on element dimensions; thus, these requirements increase by a factor of 8.

Whereas predicting the cost increase with mesh refinement for the explicit method is rather straightforward, cost is more difficult to predict when using the implicit method. The difficulty arises from the problem-dependent relationship between element connectivity and solution cost, a relationship that does not exist in the explicit method. Using the implicit method, experience shows that for many problems the computational cost is roughly proportional to the square of the number of degrees of freedom. Consider the same example of a three-dimensional model with uniform, square elements. Refining the mesh by a factor of two in all three directions increases the number of degrees of freedom by approximately 2^3, causing the computational cost to increase by a factor of roughly $(2^3)^2$, or 64. The disk space and memory requirements increase in the same manner, although the actual increase is difficult to predict.

The explicit method shows great cost savings over the implicit method as the model size increases, as long as the mesh is relatively uniform. Figure 2–16 illustrates the comparison of cost versus model size using the explicit and implicit methods. For this problem the number of degrees of freedom scales with the number of elements.

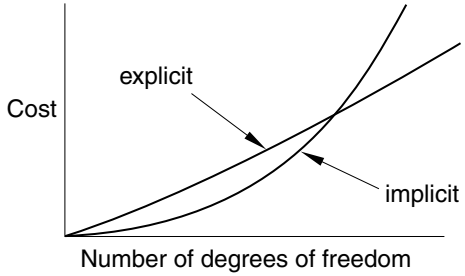


Figure 2–16 Cost versus model size in using the explicit and implicit methods.

2.5 Summary

- The Abaqus input file contains a complete description of the analysis model. It is the means of communication between the preprocessor (Abaqus/CAE, for example) and the analysis product (Abaqus/Standard or Abaqus/Explicit).
- The input file contains two sections: the model data defining the structure being analyzed and the history data defining what happens to the structure.
- Each section of the input file comprises a number of option blocks, each consisting of a keyword line, which may be followed by data lines.

- You can perform a **datacheck** analysis once you have created the input file. Error and warning messages are printed to the data file. After a successful **datacheck** analysis, estimates of the computer resources required for the simulation are printed to the data file.
- Use Abaqus/Viewer to verify the model geometry and boundary conditions graphically, using the output database file generated during the **datacheck** phase.
- It is often easiest to check for mistakes in material properties in the data (`.dat`) file; geometry, loads, and boundary conditions are more easily checked with a graphical postprocessor such as Abaqus/Viewer.
- Always check that the results satisfy basic engineering principles, such as equilibrium.
- Abaqus/Viewer allows you to visualize analysis results graphically in a variety of ways and also allows you to write tabular data reports.
- The choice between using implicit or explicit methods depends largely on the nature of the problem.

3. Finite Elements and Rigid Bodies

Finite elements and rigid bodies are the fundamental components of an Abaqus model. Finite elements are deformable, whereas rigid bodies move through space without changing shape. While users of finite element analysis programs tend to have some understanding of what finite elements are, the general concept of rigid bodies within a finite element program may be somewhat new.

For computational efficiency Abaqus has a general rigid body capability. Any body or part of a body can be defined as a rigid body; most element types can be used in a rigid body definition (the exceptions are listed in "Rigid body definition," Section 2.4.1 of the Abaqus Analysis User's Manual). The advantage of rigid bodies over deformable bodies is that the motion of a rigid body is described completely by no more than six degrees of freedom at a reference node. In contrast, deformable elements have many degrees of freedom and require expensive element calculations to determine the deformations. When such deformations are negligible or not of interest, modeling a component as a rigid body produces significant computational savings without affecting the overall results.

3.1 Finite elements

A wide range of elements is available in Abaqus. This extensive element library provides you with a powerful set of tools for solving many different problems. The elements available in Abaqus/Explicit are (with a few exceptions) a subset of those available in Abaqus/Standard. This section introduces you to the five aspects of an element that influence how it behaves.

3.1.1 Characterizing elements

Each element is characterized by the following:

- Family
- Degrees of freedom (directly related to the element family)
- Number of nodes
- Formulation
- Integration

Each element in Abaqus has a unique name, such as T2D2, S4R, or C3D8I. The element name, as you saw in the overhead hoist example in Chapter 2, "Abaqus Basics," is used as the value of the TYPE parameter on the *ELEMENT option in the input file. The element name identifies each of the five aspects of an element. The naming convention is explained in this chapter.

Family

Figure 3–1 shows the element families most commonly used in a stress analysis. One of the major distinctions between different element families is the geometry type that each family assumes.

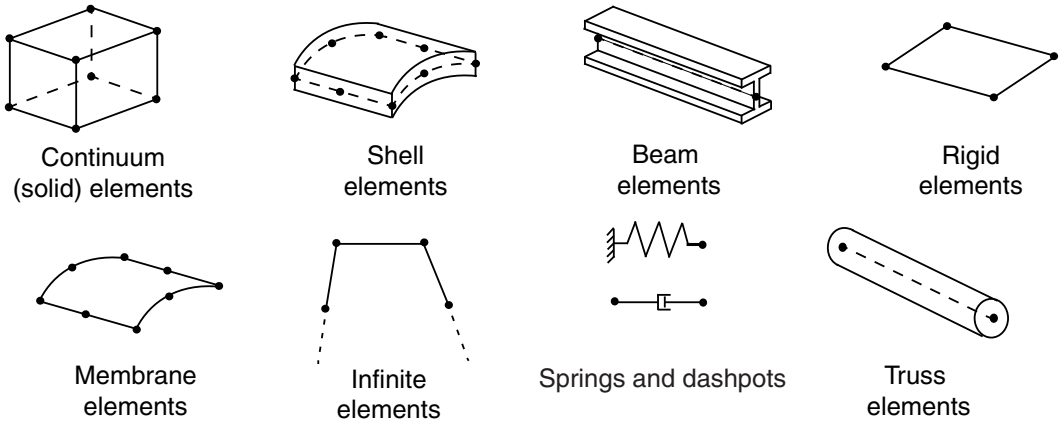


Figure 3–1 Commonly used element families.

The element families that you will use in this guide—continuum, shell, beam, truss, and rigid elements—are discussed in detail in other chapters. The other element families are not covered in this guide; if you are interested in using them in your models, read about them in Part VI, "Elements," of the Abaqus Analysis User's Manual.

The first letter or letters of an element's name indicate to which family the element belongs. For example, the S in S4R indicates this is a shell element, while the C in C3D8I indicates this is a continuum element.

Degrees of freedom

The degrees of freedom (dof) are the fundamental variables calculated during the analysis. For a stress/displacement simulation the degrees of freedom are the translations at each node. Some element families, such as the beam and shell families, have rotational degrees of freedom as well. For a heat transfer simulation the degrees of freedom are the temperatures at each node; a heat transfer analysis, therefore, requires the use of different elements than a stress analysis, since the degrees of freedom are not the same.

The following numbering convention is used for the degrees of freedom in Abaqus:

1	Translation in direction 1
2	Translation in direction 2
3	Translation in direction 3

4	Rotation about the 1-axis
5	Rotation about the 2-axis
6	Rotation about the 3-axis
7	Warping in open-section beam elements
8	Acoustic pressure, pore pressure, or hydrostatic fluid pressure
9	Electric potential
11	Temperature (or normalized concentration in mass diffusion analysis) for continuum elements or temperature at the first point through the thickness of beams and shells
12+	Temperature at other points through the thickness of beams and shells

Directions 1, 2, and 3 correspond to the global 1-, 2-, and 3-directions, respectively, unless a local coordinate system has been defined at the nodes.

Axisymmetric elements are the exception, with the displacement and rotation degrees of freedom referred to as follows:

1	Translation in the r-direction
2	Translation in the z-direction
6	Rotation in the r–z plane

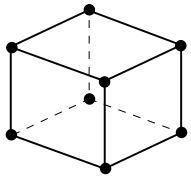
Directions r (radial) and z (axial) correspond to the global 1- and 2-directions, respectively, unless a local coordinate system has been defined at the nodes. See Chapter 5, "Using Shell Elements," for a discussion of defining a local coordinate system at the nodes.

In this guide our attention is restricted to structural applications. Therefore, only elements with translational and rotational degrees of freedom are discussed. For information on other types of elements (for example, heat transfer elements), consult the Abaqus Analysis User's Manual.

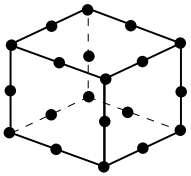
By default, Abaqus/CAE uses the alphabetical option, x-y-z, for labeling the view orientation triad. In general, this manual adopts the numerical option, 1-2-3, to permit direct correspondence with degree of freedom and output labeling. For more information on labeling of axes, see "Customizing the view triad," Section 5.4 of the Abaqus/CAE User's Manual.

Number of nodes—order of interpolation

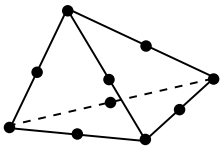
Displacements, rotations, temperatures, and the other degrees of freedom mentioned in the previous section are calculated only at the nodes of the element. At any other point in the element, the displacements are obtained by interpolating from the nodal displacements. Usually the interpolation order is determined by the number of nodes used in the element, as illustrated in the examples in Figure 3–2.



(a) Linear element
(8-node brick, C3D8)



(b) Quadratic element
(20-node brick, C3D20)



(c) Modified second-order element
(10-node tetrahedron, C3D10M)

Figure 3–2 Linear brick, quadratic brick, and modified tetrahedral elements.

- Elements that have nodes only at their corners, such as the 8-node brick shown in Figure 3–2(a), use linear interpolation in each direction and are often called linear elements or first-order elements.
- Elements with midside nodes, such as the 20-node brick shown in Figure 3–2(b), use quadratic interpolation and are often called quadratic elements or second-order elements.
- Modified triangular or tetrahedral elements with midside nodes, such as the 10-node tetrahedron shown in Figure 3–2(c), use a modified second-order interpolation and are often called modified elements or modified second-order elements.

Abaqus/Standard offers a wide selection of both linear and quadratic elements. Abaqus/Explicit offers only linear elements, with the exception of the quadratic beam and modified tetrahedron and triangle elements.

Typically, the number of nodes in an element is clearly identified in its name. The 8-node brick element, as you have seen, is called C3D8; and the 8-node general shell element is called S8R. The beam element family uses a slightly different convention: the order of interpolation is identified in the name. Thus, a first-order, three-dimensional beam element is called B31, whereas a second-order, three-dimensional beam element is called B32. A similar convention is used for axisymmetric shell and membrane elements.

Formulation

An element's formulation refers to the mathematical theory used to define the element's behavior. In the absence of adaptive meshing all of the stress/displacement elements in Abaqus are based on the *Lagrangian* or *material* description of behavior: the material associated with an element remains associated with the element throughout the analysis, and material cannot flow across element boundaries. In the alternative *Eulerian* or *spatial* description, elements are fixed in space as the material flows through them. Eulerian methods are used commonly in fluid mechanics simulations. Abaqus/Standard uses Eulerian elements to model convective heat transfer. Adaptive meshing combines the features of pure Lagrangian and Eulerian analyses and allows the motion of the element to be independent of the material. Eulerian elements and adaptive meshing are not discussed in this guide.

To accommodate different types of behavior, some element families in Abaqus include elements with several different formulations. For example, the shell element family has three classes: one suitable for general-purpose shell analysis, another for thin shells, and yet another for thick shells. (These shell element formulations are explained in Chapter 5, "Using Shell Elements.")

Some Abaqus/Standard element families have a standard formulation as well as some alternative formulations. Elements with alternative formulations are identified by an additional character at the end of the element name. For example, the continuum, beam, and truss element families include members with a hybrid formulation in which the pressure (continuum elements) or axial force (beam and truss elements) is treated as an additional unknown; these elements are identified by the letter "H" at the end of the name (C3D8H or B31H).

Some element formulations allow coupled field problems to be solved. For example, elements whose names begin with the letter C and end with the letter T (such as C3D8T) possess both mechanical and thermal degrees of freedom and are intended for coupled thermal-mechanical simulations.

Several of the most commonly used element formulations are discussed later in this guide.

Integration

Abaqus uses numerical techniques to integrate various quantities over the volume of each element. Using Gaussian quadrature for most elements, Abaqus evaluates the material response at each integration point in each element. Some elements in Abaqus can use full or reduced integration, a choice that can have a significant effect on the accuracy of the element for a given problem, as discussed in detail in "Element formulation and integration," Section 4.1.

Abaqus uses the letter "R" at the end of the element name to distinguish reduced-integration elements (unless they are also hybrid elements, in which case the element name ends with the letters "RH"). For example, CAX4 is the 4-node, fully integrated, linear, axisymmetric solid element; and CAX4R is the reduced-integration version of the same element.

Abaqus/Standard offers both full and reduced-integration elements; Abaqus/Explicit offers only reduced-integration elements with the exception of the modified tetrahedron and triangle elements and the fully integrated first-order shell, membrane, and brick elements.

3.1.2 Continuum elements

Among the different element families, continuum or solid elements can be used to model the widest variety of components. Conceptually, continuum elements simply model small blocks of material in a component. Since they may be connected to other elements on any of their faces, continuum elements, like bricks in a building or tiles in a mosaic, can be used to build models of nearly any shape, subjected to nearly any loading. Abaqus has stress/displacement, nonstructural, and coupled field continuum elements; this guide will discuss only stress/displacement elements.

Continuum stress/displacement elements in Abaqus have names that begin with the letter "C." The next two letters indicate the dimensionality and usually, but not always, the active degrees of freedom

in the element. The letters “3D” indicate a three-dimensional element; “AX,” an axisymmetric element; “PE,” a plane strain element; and “PS,” a plane stress element.

The use of continuum elements is discussed further in Chapter 4, “Using Continuum Elements.”

Three-dimensional continuum element library

Three-dimensional continuum elements can be hexahedra (bricks), wedges, or tetrahedra. The full inventory of three-dimensional continuum elements and the nodal connectivity for each type can be found in “Three-dimensional solid element library,” Section 28.1.4 of the Abaqus Analysis User’s Manual.

Whenever possible, hexahedral elements or second-order tetrahedral elements should be used in Abaqus. First-order tetrahedra (C3D4) have a simple, constant-strain formulation, and very fine meshes are required for an accurate solution.

Two-dimensional continuum element library

Abaqus has several classes of two-dimensional continuum elements that differ from each other in their out-of-plane behavior. Two-dimensional elements can be quadrilateral or triangular. Figure 3–3 shows the three classes that are used most commonly.

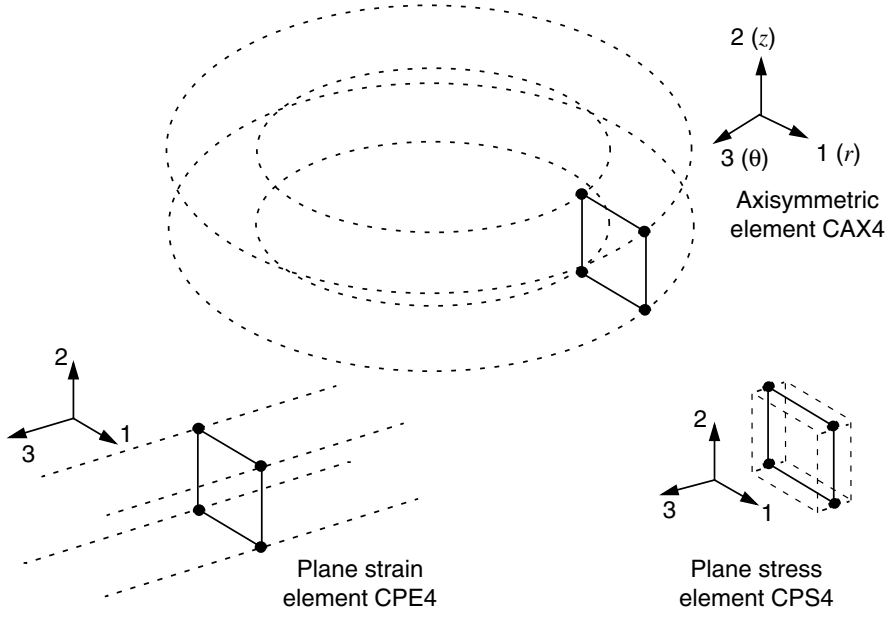


Figure 3–3 Plane strain, plane stress, and axisymmetric elements without twist.

Plane strain elements assume that the out-of-plane strain, ε_{33}, is zero; they can be used to model thick structures.

Plane stress elements assume that the out-of-plane stress, σ_{33}, is zero; they are suitable for modeling thin structures.

Axisymmetric elements without twist, the "CAX" class of elements, model a 360° ring; they are suitable for analyzing structures with axisymmetric geometry subjected to axisymmetric loading.

Abaqus/Standard also provides generalized plane strain elements, axisymmetric elements with twist, and axisymmetric elements with asymmetric deformation.

- Generalized plane strain elements include the additional generalization that the out-of-plane strain may vary linearly with position in the plane of the model. This formulation is particularly suited for the thermal-stress analysis of thick sections.
- Axisymmetric elements with twist model an initially axisymmetric geometry that can twist about the axis of symmetry. These elements are useful for modeling the torsion of cylindrical structures, such as axisymmetric rubber bushings.
- Axisymmetric elements with asymmetric deformation model an initially axisymmetric geometry that can deform asymmetrically (typically as a result of bending). They are useful for simulating problems such as an axisymmetric rubber mount that is subjected to shear loads.

The latter three classes of two-dimensional continuum elements are not discussed in this guide.

Two-dimensional solid elements must be defined in the 1–2 plane so that the node order is counterclockwise around the element perimeter, as shown in Figure 3–4.

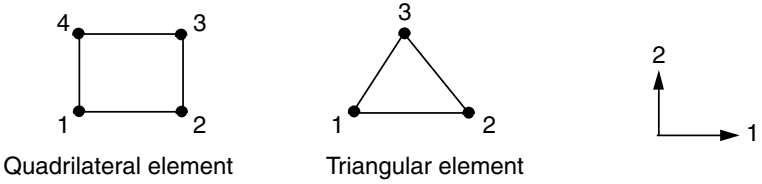


Figure 3–4 Correct nodal connectivity for two-dimensional elements.

When using a preprocessor to generate the mesh, ensure that the element normals all point in the same direction as the positive, global 3-axis. Failure to provide the correct element connectivity will cause Abaqus to issue an error message stating that elements have negative area.

Degrees of freedom

All of the stress/displacement continuum elements have translational degrees of freedom at each node. Correspondingly, degrees of freedom 1, 2, and 3 are active in three-dimensional elements, while only degrees of freedom 1 and 2 are active in plane strain elements, plane stress elements, and axisymmetric elements without twist. To find the active degrees of freedom in the other classes

of two-dimensional solid elements, see "Two-dimensional solid element library," Section 28.1.3 of the Abaqus Analysis User's Manual.

Element properties

The *SOLID SECTION option defines the material and any additional geometric data associated with a set of continuum elements. For three-dimensional and axisymmetric elements no additional geometric information is required: the nodal coordinates completely define the element geometry. For plane stress and plane strain elements the thickness of the elements must be specified on the data line. For example, if the elements are 0.2 m thick, the element property definition would be the following:

```
*SOLID SECTION, ELSET=<element set name>, MATERIAL=<material name>
0.2,
```

Formulation and integration

Alternative formulations available for the continuum family of elements in Abaqus/Standard include an *incompatible mode* formulation (the last or second-to-last letter in the element name is I) and a *hybrid* element formulation (the last letter in the element name is H), both of which are discussed in detail later in this guide.

In Abaqus/Standard you can choose between full and reduced integration for quadrilateral and hexahedral (brick) elements. In Abaqus/Explicit you can choose between full and reduced integration for hexahedral (brick) elements; however, only reduced integration is available for quadrilateral first-order elements. Both the formulation and type of integration can have a significant effect on the accuracy of solid elements, as discussed in "Element formulation and integration," Section 4.1.

Element output variables

By default, element output variables such as stress and strain refer to the global Cartesian coordinate system. Thus, the σ_{11}-component of stress at the integration point shown in Figure 3–5(a) acts in the global 1-direction. Even if the element rotates during a large-displacement simulation, as shown in Figure 3–5(b), the default is still to use the global Cartesian system as the basis for defining the element variables. However, Abaqus allows you to define a local coordinate system for element variables (see "Example: skew plate," Section 5.5). This local coordinate system rotates with the motion of the element in large-displacement simulations. A local coordinate system can be very useful if the object being modeled has some natural material orientation, such as the fiber directions in a composite material.

3.1.3 Shell elements

Shell elements are used to model structures in which one dimension (the thickness) is significantly smaller than the other dimensions and the stresses in the thickness direction are negligible.

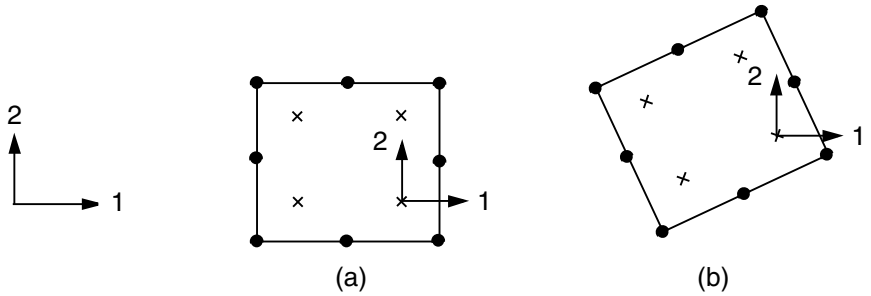


Figure 3–5 Default material directions for continuum elements.

Shell element names in Abaqus begin with the letter "S." Axisymmetric shells all begin with the letters "SAX." Abaqus/Standard also provides axisymmetric shells with asymmetric deformations, which begin with the letters "SAXA." The first number in a shell element name indicates the number of nodes in the element, except for the case of axisymmetric shells, for which the first number indicates the order of interpolation.

Two types of shell elements are available in Abaqus: conventional shell elements and continuum shell elements. Conventional shell elements discretize a reference surface by defining the element's planar dimensions, its surface normal, and its initial curvature. Continuum shell elements, on the other hand, resemble three-dimensional solid elements in that they discretize an entire three-dimensional body yet are formulated so that their kinematic and constitutive behavior is similar to conventional shell elements. In this manual only conventional shell elements are discussed. Henceforth, we will refer to them simply as "shell elements." For more information on continuum shell elements, see "Shell elements: overview," Section 29.6.1 of the Abaqus Analysis User's Manual.

The use of shell elements is discussed in detail in Chapter 5, "Using Shell Elements."

Shell element library

In Abaqus/Standard general three-dimensional shell elements are available with three different formulations: general-purpose, thin-only, and thick-only. The general-purpose shells and the axisymmetric shells with asymmetric deformation account for finite membrane strains and arbitrarily large rotations. The three-dimensional "thick" and "thin" element types provide for arbitrarily large rotations but only small strains. The general-purpose shells allow the shell thickness to change with the element deformation. All of the other shell elements assume small strains and no change in shell thickness, even though the element's nodes may undergo finite rotations. Triangular and quadrilateral elements with linear and quadratic interpolation are available. Both linear and quadratic axisymmetric shell elements are available. All of the quadrilateral shell elements (except for S4) and the triangular shell element S3/S3R use reduced integration. The S4 element and the other triangular shell elements use full integration. Table 3–1 summarizes the shell elements available in Abaqus/Standard.

All the shell elements in Abaqus/Explicit are general-purpose. Finite membrane strain and small membrane strain formulations are available. Triangular and quadrilateral elements are

Table 3–1 Three classes of shell elements in Abaqus/Standard.

General-Purpose Shells	Thin-Only Shells	Thick-Only Shells
S4, S4R, S3/S3R, SAX1, SAX2, SAX2T, SC6R, SC8R	STRI3, STRI65, S4R5, S8R5, S9R5, SAXA	S8R, S8RT

available with linear interpolation. A linear axisymmetric shell element is also available. Table 3–2 summarizes the shell elements available in Abaqus/Explicit.

Table 3–2 Two classes of shell elements in Abaqus/Explicit.

Finite-Strain Shells	Small-Strain Shells
S4, S4R, S3/S3R, SAX1	S4RS, S4RSW, S3RS

For most explicit analyses the large-strain shell elements are appropriate. If, however, the analysis involves small membrane strains and arbitrarily large rotations, the small-strain shell elements are more computationally efficient. The S4RS and S3RS elements do not consider warping, while the S4RSW element does.

The shell formulations available in Abaqus are discussed in detail in Chapter 5, "Using Shell Elements."

Degrees of freedom

The three-dimensional elements in Abaqus/Standard whose names end in the number "5" (e.g., S4R5, STRI65) have 5 degrees of freedom at each node: three translations and two in-plane rotations (i.e., no rotations about the shell normal). However, all six degrees of freedom are activated at a node if required; for example, if rotational boundary conditions are applied or if the node is on a fold line of the shell.

The remaining three-dimensional shell elements have six degrees of freedom at each node (three translations and three rotations).

The axisymmetric shells have three degrees of freedom associated with each node:

1 Translation in the r-direction.

2 Translation in the z-direction.

6 Rotation in the r–z plane.

Element properties

Use either the *SHELL GENERAL SECTION or the *SHELL SECTION option to define the thickness and material properties for a set of shell elements. These two options have similar formats:

***SHELL SECTION, ELSET=**<*element set name*>**, MATERIAL=**<*material name*>
<*thickness*>**,**<*number of section points*>

or

***SHELL GENERAL SECTION, ELSET=**<*element set name*>**,**
MATERIAL=<*material name*>
<*thickness*>

If you specify the ∗SHELL SECTION option, Abaqus uses numerical integration to calculate the behavior at selected points through the thickness of the shell. These points are called section points, as shown in Figure 3–6. The MATERIAL parameter refers to a material property definition, which may be linear or nonlinear. You can specify any odd number of section points through the shell thickness.

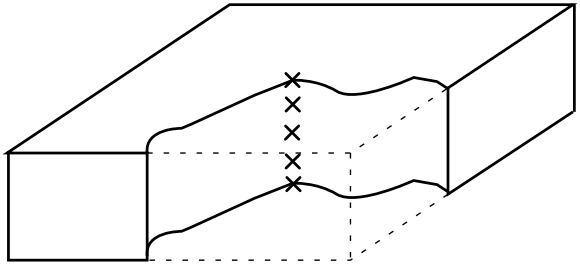


Figure 3–6 Section points through the thickness of a shell element.

The ∗SHELL GENERAL SECTION option allows you to define the cross-section behavior in a number of general ways to model linear or nonlinear behavior. Since Abaqus models the shell's cross-section behavior directly in terms of section engineering quantities (area, moments of inertia, etc.) with this option, there is no need for Abaqus to integrate any quantities over the element cross-section. Therefore, ∗SHELL GENERAL SECTION is less expensive computationally than ∗SHELL SECTION. The response is calculated in terms of force and moment resultants; the stresses and strains are calculated only when they are requested for output.

Reference surface offsets

The reference surface of the shell is defined by the shell element's nodes and normal definitions. When modeling with shell elements, the reference surface is typically coincident with the shell's midsurface. However, many situations arise in which it is more convenient to define the reference surface as offset from the shell's midsurface. For example, surfaces created in CAD packages usually represent either the top or bottom surface of the shell body. In this case it may be easier to define the reference surface to be coincident with the CAD surface and, therefore, offset from the shell's midsurface.

Shell offsets can also be used to define a more precise surface geometry for contact problems where shell thickness is important. By default, shell offset and thickness are accounted for in contact constraints in Abaqus/Explicit. The effect of offset and thickness in contact can be suppressed, if required.

Shell offsets can also be useful when modeling a shell with continuously varying thickness. In this case defining the nodes at the shell midplane can be difficult. If one surface is smooth while the other is rough, as in some aircraft structures, it is easiest to use shell offsets to define the nodes at the smooth surface.

Offsets can be introduced by using the OFFSET parameter on the *SHELL SECTION and *SHELL GENERAL SECTION options. The offset value is defined as a fraction of the shell thickness measured from the shell's midsurface to the shell's reference surface.

The degrees of freedom for the shell are associated with the reference surface. All kinematic quantities, including the element's area, are calculated there. Large offset values for curved shells may lead to a surface integration error, affecting the stiffness, mass, and rotary inertia for the shell section. For stability purposes Abaqus/Explicit also automatically augments the rotary inertia used for shell elements on the order of the offset squared, which may result in errors in the dynamics for large offsets. When large offsets from the shell's midsurface are necessary, use multi-point constraints or rigid body constraints instead.

Element output variables

The element output variables for shells are defined in terms of local material directions that lie on the surface of each shell element. In all large-displacement simulations these axes rotate with the element's deformation. You can also define a local material coordinate system that rotates with the element's deformation in a large-displacement analysis.

3.1.4 Beam elements

Beam elements are used to model components in which one dimension (the length) is significantly greater than the other two dimensions and only the stress in the direction along the axis of the beam is significant.

Beam element names in Abaqus begin with the letter "B." The next character indicates the dimensionality of the element: "2" for two-dimensional beams and "3" for three-dimensional beams. The third character indicates the interpolation used: "1" for linear interpolation, "2" for quadratic interpolation, and "3" for cubic interpolation.

The use of beam elements is discussed in Chapter 6, "Using Beam Elements."

Beam element library

Linear, quadratic, and cubic beams are available in two and three dimensions. Cubic beams are not available in Abaqus/Explicit.

Degrees of freedom

Three-dimensional beams have six degrees of freedom at each node: three translational degrees of freedom (1–3) and three rotational degrees of freedom (4–6). "Open-section"-type beams (such as B31OS) are available in Abaqus/Standard and have an additional degree of freedom (7) that represents the warping of the beam cross-section.

Two-dimensional beams have three degrees of freedom at each node: two translational degrees of freedom (1 and 2) and one rotational degree of freedom (6) about the normal to the plane of the model.

Element properties

Use either the *BEAM SECTION or the *BEAM GENERAL SECTION option to define the geometry of the beam cross-section; the nodal coordinates define only the length.

If you specify the *BEAM SECTION option, the beam cross-section is defined geometrically, and the MATERIAL parameter refers to a material property definition. Abaqus calculates the cross-section behavior of the beam by numerical integration over the cross-section, allowing both linear and nonlinear material behavior.

The *BEAM GENERAL SECTION option allows you to define the cross-section behavior in a number of general ways to model linear or nonlinear behavior. Since Abaqus models the beam's cross-section behavior directly in terms of section engineering quantities (area, moments of inertia, etc.) with this option, there is no need for Abaqus to integrate any quantities over the element cross-section. Therefore, *BEAM GENERAL SECTION is less expensive computationally than *BEAM SECTION. The response is calculated in terms of the force and moment resultants; the stresses and strains are calculated only when they are requested for output.

In Abaqus/Standard you can also define beams with linearly tapered cross-sections. General beam sections with linear response and standard library sections are supported.

Formulation and integration

The linear beams (B21 and B31) and the quadratic beams (B22 and B32) are shear deformable and account for finite axial strains; therefore, they are suitable for modeling both slender and stout beams. The cubic beam elements in Abaqus/Standard (B23 and B33) do not account for shear flexibility and assume small axial strain, although large displacements and rotations of the beams are valid. They are, therefore, suitable for modeling slender beams.

Abaqus/Standard provides variants of linear and quadratic beam elements that are suitable for modeling thin-walled, open-section beams (B31OS and B32OS). These elements model the effects of torsion and warping in open cross-sections, such as I-beams or U-section channels. Open-section beams are not covered in this guide.

Abaqus/Standard also has hybrid beam elements that are used for modeling very slender members, such as flexible risers on offshore oil installations, or for modeling very stiff links. Hybrid beams are not covered in this guide.

Element output variables

The stress components in three-dimensional, shear-deformable beam elements are the axial stress (σ_{11}) and the shear stress due to torsion (σ_{12}). The shear stress acts about the section wall in a thin-walled section. Corresponding strain measures are also available. The shear-deformable beams also provide estimates of transverse shear forces on the section. The slender (cubic) beams in Abaqus/Standard have only the axial variables as output. Open-section beams in space also have only the axial variables as output, since the torsional shear stresses are negligible in this case.

All two-dimensional beams use only axial stress and strain.

The axial force, bending moments, and curvatures about the local beam axes can also be requested for output. For details of what components are available with which elements, see "Beam modeling: overview," Section 29.3.1 of the Abaqus Analysis User's Manual. Details of how the local beam axes are defined are given in Chapter 6, "Using Beam Elements."

3.1.5 Truss elements

Truss elements are rods that can carry only tensile or compressive loads. They have no resistance to bending; therefore, they are useful for modeling pin-jointed frames. Moreover, truss elements can be used as an approximation for cables or strings (for example, in a tennis racket). Trusses are also sometimes used to represent reinforcement within other elements. The overhead hoist model in Chapter 2, "Abaqus Basics," uses truss elements.

All truss element names begin with the letter "T." The next two characters indicate the dimensionality of the element—"2D" for two-dimensional trusses and "3D" for three-dimensional trusses. The final character represents the number of nodes in the element.

Truss element library

Linear and quadratic trusses are available in two and three dimensions. Quadratic trusses are not available in Abaqus/Explicit.

Degrees of freedom

Truss elements have only translational degrees of freedom at each node. Three-dimensional truss elements have degrees of freedom 1, 2, and 3, while two-dimensional truss elements have degrees of freedom 1 and 2.

Element properties

The *SOLID SECTION option is used to specify the name of the material property definition associated with the given set of truss elements. The cross-sectional area is given on the data line:

***SOLID SECTION, ELSET=**<*element set name*>**, MATERIAL=**<*material*>
<*cross-sectional area*>

Formulation and integration

In addition to the standard formulation, a hybrid truss element formulation is available in Abaqus/Standard. It is useful for modeling very stiff links whose stiffness is much greater than that of the overall structure.

Element output variables

Axial stress and strain are available as output for truss elements.

3.2 Rigid bodies

In Abaqus a rigid body is a collection of nodes and elements whose motion is governed by the motion of a single node, known as the *rigid body reference node*, as shown in Figure 3–7.

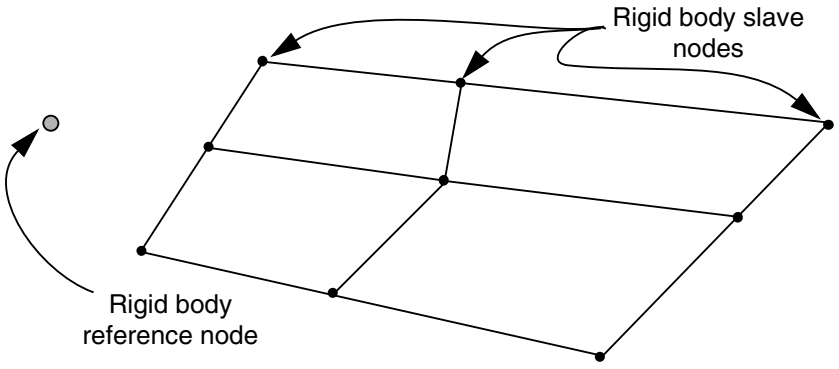


Figure 3–7 Elements forming a rigid body.

The shape of the rigid body is defined either as an analytical surface obtained by revolving or extruding a two-dimensional geometric profile or as a discrete rigid body obtained by meshing the body with nodes and elements. The shape of the rigid body does not change during a simulation but can undergo large rigid body motions. The mass and inertia of a discrete rigid body can be calculated based on the contributions from its elements, or they can be assigned directly.

The motion of a rigid body can be prescribed by applying boundary conditions at the rigid body reference node. Loads on a rigid body are generated from concentrated loads applied to nodes and

distributed loads applied to elements that are part of the rigid body or from loads applied to the rigid body reference node. Rigid bodies interact with the rest of the model through nodal connections to deformable elements and through contact with deformable elements.

The use of rigid bodies is illustrated in Chapter 12, “Contact.”

3.2.1 Determining when to use a rigid body

Rigid bodies can be used to model very stiff components that are either fixed or undergoing large rigid body motions. They can also be used to model constraints between deformable components, and they provide a convenient method of specifying certain contact interactions. When Abaqus is used for quasi-static forming analyses, rigid bodies are ideally suited for modeling tooling (such as a punch, die, drawbead, blank holder, roller, etc.) and may also be effective as a method of constraint.

It may be useful to make parts of a model rigid for verification purposes. For example, in complex models elements far away from the particular region of interest could be included as part of a rigid body, resulting in faster run times at the model development stage. When you are satisfied with the model, you can remove the rigid body definitions and incorporate an accurate deformable finite element representation throughout.

The principal advantage to representing portions of a model with rigid bodies rather than deformable finite elements is computational efficiency. Element-level calculations are not performed for elements that are part of a rigid body. Although some computational effort is required to update the motion of the nodes of the rigid body and to assemble concentrated and distributed loads, the motion of the rigid body is determined completely by a maximum of six degrees of freedom at the rigid body reference node.

In Abaqus/Explicit rigid bodies are particularly effective for modeling relatively stiff parts of a structure for which tracking stress waves and distributions is not important. Element stable time increment estimates in the stiff region can result in a very small global time increment. Since rigid bodies and elements that are part of a rigid body do not affect the global time increment, using a rigid body instead of a deformable finite element representation in a stiff region can result in a much larger global time increment, without significantly affecting the overall accuracy of the solution.

Rigid bodies defined with analytical rigid surfaces in Abaqus are slightly cheaper in terms of computational cost than discrete rigid bodies and may yield smoother results. In Abaqus/Explicit, for example, contact with analytical rigid surfaces tends to be less noisy than contact with discrete rigid bodies because analytical rigid surfaces can be smooth, whereas discrete rigid bodies are inherently faceted. However, the shapes that can be defined with analytical rigid surfaces are limited.

3.2.2 Components of a rigid body

To create a discrete rigid body, use the *RIGID BODY option as the property reference for the elements forming the rigid body. Use the REF NODE parameter to assign a rigid body reference node to the rigid body. A rigid body reference node has both translational and rotational degrees of freedom and must be defined for every rigid body. The position of the rigid body reference node is not important unless rotations are applied to the body or reaction moments about a certain axis through the body are desired.

In either of these situations the node should be placed such that it lies on the desired axis through the body.

***RIGID BODY, REF NODE=**<*node*>**, ELSET=**<*element set name*>**,**
PIN NSET=<*node set name*>**, TIE NSET=**<*node set name*>

In addition to the rigid body reference node, discrete rigid bodies consist of a collection of nodes that are generated by assigning elements and nodes to the rigid body. These nodes, known as the *rigid body slave nodes* (see Figure 3–7), provide a connection to other elements. Nodes that are part of a rigid body are one of two types:

- Pin nodes, which have only translational degrees of freedom.
- Tie nodes, which have both translational and rotational degrees of freedom.

The rigid body node type is determined by the type of elements on the rigid body to which the node is attached. The node type also can be specified or modified when assigning nodes directly to a rigid body. For pin nodes only the translational degrees of freedom are part of the rigid body, and the motion of these degrees of freedom is constrained by the motion of the rigid body reference node. For tie nodes both the translational and rotational degrees of freedom are part of the rigid body and are constrained by the motion of the rigid body reference node.

The nodes defining the rigid body cannot have any boundary conditions, multi-point constraints, or constraint equations applied to them. Boundary conditions, multi-point constraints, constraint equations, and loads can be applied, however, to the rigid body reference node.

3.2.3 Rigid elements

The rigid body capability in Abaqus allows most elements—not just rigid elements—to be part of a rigid body. For example, shell elements or rigid elements can be used to model the same effect if the ∗RIGID BODY option refers to the element set that contains the elements forming the rigid body. The rules governing rigid bodies, such as how loads and boundary conditions are applied, pertain to all element types that form the rigid body, including rigid elements.

The names of all rigid elements begin with the letter “R.” The next characters indicate the dimensionality of the element. For example, “2D” indicates that the element is planar; and “AX,” that the element is axisymmetric. The final character represents the number of nodes in the element.

Rigid element library

The three-dimensional quadrilateral (R3D4) and triangular (R3D3) rigid elements are used to model the two-dimensional surfaces of a three-dimensional rigid body. Another element—a two-node, rigid beam element (RB3D2)—is provided in Abaqus/Standard mainly to model components of offshore structures to which fluid drag and buoyancy loads must be applied.

Two-node, rigid elements are available for plane strain, plane stress, and axisymmetric models. A planar, two-node rigid beam element is also available in Abaqus/Standard and is used mainly to model offshore structures in two dimensions.

Degrees of freedom

Only the rigid body reference node has independent degrees of freedom. For three-dimensional elements the reference node has three translational and three rotational degrees of freedom; for planar and axisymmetric elements the reference node has degrees of freedom 1, 2, and 6 (rotation about the 3-axis).

The nodes attached to rigid elements have only slave degrees of freedom. The motion of these nodes is determined entirely by the motion of the rigid body reference node. For planar and three-dimensional rigid elements the only slave degrees of freedom are translations. The rigid beam elements in Abaqus/Standard have the same slave degrees of freedom as the corresponding deformable beam elements: 1–6 for the three-dimensional rigid beam and 1, 2, and 6 for the planar rigid beam.

Physical properties

All rigid elements must reference a *RIGID BODY option. For the planar and beam elements the cross-sectional area can be defined on the data line. For the axisymmetric and three-dimensional elements the thickness can be defined on the data line; these data are required only if you apply body forces to the rigid elements. The default thickness is zero. Alternatively, the NODAL THICKNESS parameter defines an average facet thickness based on the thickness at the nodes. These data are required when applying body forces or when the thickness is needed for the contact definition.

Formulation and integration

Since the rigid elements are not deformable, they do not use numerical integration points, and there are no optional formulations.

Element output variables

There are no element output variables. The only output from rigid elements is the motion of the nodes. In addition, reaction forces and reaction moments are available at the rigid body reference node.

3.3 Summary

- Abaqus has an extensive library of elements that can be used for a wide range of structural applications. Your choice of element type has important consequences regarding the accuracy and efficiency of your simulation. The elements available in Abaqus/Explicit are (in general) a subset of those available in Abaqus/Standard.
- The degrees of freedom active at a node depend on the element types attached to the node.

- The element name completely identifies the element's family, formulation, number of nodes, and type of integration.
- All elements must refer to a section property definition. The section property provides any additional data required to define the geometry of the element and also identifies the associated material property definition.
- For continuum elements Abaqus defines the element output variables, such as stress and strain, with respect to the global Cartesian coordinate system. You can change to a local coordinate system by using the *ORIENTATION option.
- For three-dimensional shell elements Abaqus defines the element output variables with respect to a coordinate system based on the surface of the shell. You can change the coordinate system by using the *ORIENTATION option.
- For computational efficiency any part of a model can be defined as a rigid body, which has degrees of freedom only at its reference node.
- As a method of constraint in an Abaqus/Explicit analysis, rigid bodies are computationally more efficient than multi-point constraints.

4. Using Continuum Elements

The continuum (solid) family of stress/displacement elements is the most comprehensive of the element libraries in Abaqus. There are some differences in the solid element libraries available in Abaqus/Standard and Abaqus/Explicit.

Abaqus/Standard solid element library

The Abaqus/Standard solid element library includes first-order (linear) interpolation elements and second-order (quadratic) interpolation elements in two or three dimensions using either full or reduced integration. Triangles and quadrilaterals are available in two dimensions; and tetrahedra, triangular wedges, and hexahedra ("bricks") are provided in three dimensions. Modified second-order triangular and tetrahedral elements are also provided.

In addition, hybrid and incompatible mode elements are available in Abaqus/Standard.

Abaqus/Explicit solid element library

The Abaqus/Explicit solid element library includes reduced-integration first-order (linear) interpolation elements in two or three dimensions. Modified second-order interpolation triangles and tetrahedra are also available. Full integration or regular second-order elements are not available in Abaqus/Explicit, with the exception of the fully integrated first-order hexahedral element (an incompatible mode version of this element is also available).

For detailed information on the options available for continuum elements, please see "Solid (continuum) elements," Section 28.1.1 of the Abaqus Analysis User's Manual.

When the permutations of all these various element options are made, the total number of solid elements available to you is large—over 20 just for three-dimensional models. The accuracy of your simulation will depend strongly on the type of element you use in your model. The thought of choosing which of these elements is best for your model may seem daunting, especially at first. However, you will come to view this selection as a 20+ piece tool set that provides you with the ability to choose just the right tool, or element, for a particular job.

This chapter discusses the effect that different element formulations and levels of integration have on the accuracy of a particular analysis. Some general guidelines for selecting continuum elements are also given. These provide the foundation upon which you can build your knowledge as you gain more experience using Abaqus. The example at the end of this section will allow you to put this knowledge to use as you build and analyze a connecting lug.

4.1 Element formulation and integration

The influence that the order of the element (linear or quadratic), the element formulation, and the level of integration have on the accuracy of a structural simulation will be demonstrated by considering a static analysis of the cantilever beam shown in Figure 4–1.

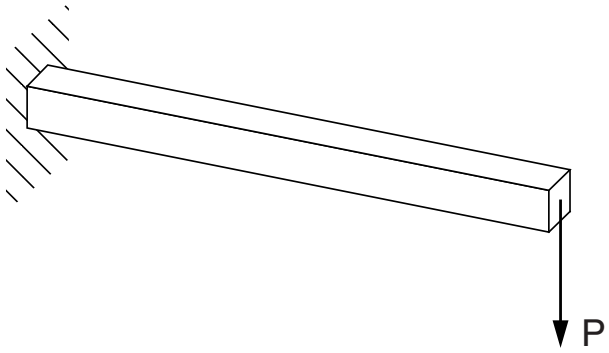


Figure 4–1 Cantilever beam under a point load P at its free end.

This is a classic test used to assess the behavior of a given finite element. Since the beam is relatively slender, we would normally model it with beam elements. However, it is used here to help assess the effectiveness of various solid elements.

The beam is 150 mm long, 2.5 mm wide, and 5 mm deep; built-in at one end; and carrying a tip load of 5 N at the free end. The material has a Young's modulus, E, of 70 GPa and a Poisson's ratio of 0.0. Using beam theory, the static deflection of the tip of the beam for a load P is given as

$$\delta_{tip} = \frac{Pl^3}{3EI},$$

where $I = bd^3/12$, l is the length, b is the width, and d is the depth of the beam.

For $P = 5$ N the tip deflection is 3.09 mm.

4.1.1 Full integration

The expression "full integration" refers to the number of Gauss points required to integrate the polynomial terms in an element's stiffness matrix exactly when the element has a regular shape. For hexahedral and quadrilateral elements a "regular shape" means that the edges are straight and meet at right angles and that any edge nodes are at the midpoint of the edge. Fully integrated, linear elements use two integration points in each direction. Thus, the three-dimensional element C3D8 uses a 2 × 2 × 2 array of integration points in the element. Fully integrated, quadratic elements (available only in Abaqus/Standard) use three integration points in each direction. The locations of the integration points in fully integrated, two-dimensional, quadrilateral elements are shown in Figure 4–2.

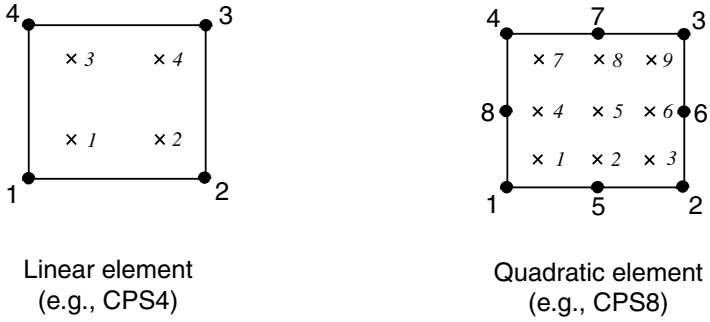


Figure 4–2 Integration points in fully integrated, two-dimensional, quadrilateral elements.

Several different finite element meshes were used in Abaqus/Standard simulations of the cantilever beam problem, as shown in Figure 4–3. The simulations use either linear or quadratic, fully integrated elements and illustrate the effects of both the order of the element (first versus second) and the mesh density on the accuracy of the results.

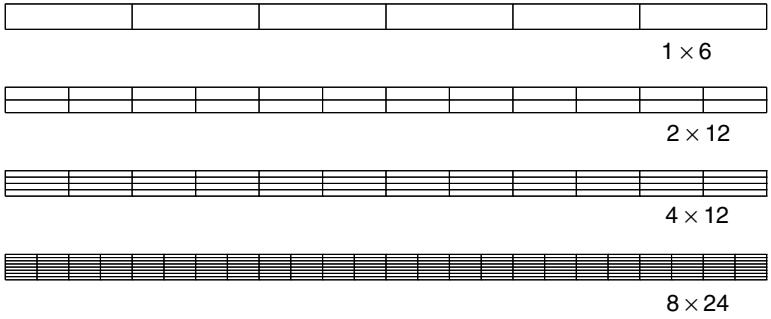


Figure 4–3 Meshes used for the cantilever beam simulations.

The ratios of the tip displacements for the various simulations to the beam-theory value of 3.09 mm are shown in Table 4–1.

The linear elements CPS4 and C3D8 underpredict the deflection so badly that the results are unusable. The results are least accurate with coarse meshes, but even a fine mesh (8 × 24) still predicts a tip displacement that is only 56% of the theoretical value. Notice that for the linear, fully integrated elements it makes no difference how many elements there are through the thickness of the beam. The underprediction of tip deflection is caused by *shear locking*, which is a problem with all fully integrated, first-order, solid elements.

Table 4–1 Normalized tip displacements with fully-integrated elements.

Element	Mesh Size (Depth × Length)			
	1 × 6	2 × 12	4 × 12	8 × 24
CPS4	0.074	0.242	0.242	0.561
CPS8	0.994	1.000	1.000	1.000
C3D8	0.077	0.248	0.243	0.563
C3D20	0.994	1.000	1.000	1.000

As we have seen, shear locking causes the elements to be too stiff in bending. It is explained as follows. Consider a small piece of material in a structure subject to pure bending. The material will distort as shown in Figure 4–4.

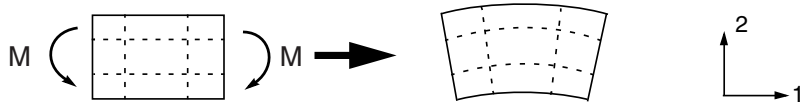


Figure 4–4 Deformation of material subjected to bending moment M.

Lines initially parallel to the horizontal axis take on constant curvature, and lines through the thickness remain straight. The angle between the horizontal and vertical lines remains at 90°.

The edges of a linear element are unable to curve; therefore, if the small piece of material is modeled using a single element, its deformed shape is like that shown in Figure 4–5.

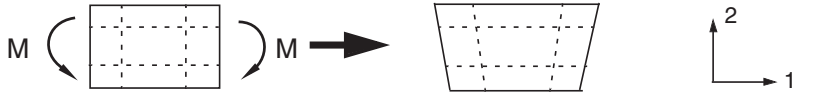


Figure 4–5 Deformation of a fully integrated, linear element subjected to bending moment M.

For visualization, dotted lines that pass through the integration points are plotted. It is apparent that the upper line has increased in length, indicating that the direct stress in the 1-direction, σ_{11}, is tensile. Similarly, the length of the lower dotted line has decreased, indicating that σ_{11} is compressive. The length of the vertical dotted lines has not changed (assuming that displacements are small); therefore, σ_{22} at all integration points is zero. All this is consistent with the expected state of stress of a small piece of material subjected to pure bending. However, at each integration point the angle between the vertical and horizontal lines, which was initially 90°, has changed. This indicates that the shear stress, σ_{12}, at

these points is nonzero. This is incorrect: the shear stress in a piece of material under pure bending is zero.

This spurious shear stress arises because the edges of the element are unable to curve. Its presence means that strain energy is creating shearing deformation rather than the intended bending deformation, so the overall deflections are less: the element is too stiff.

Shear locking only affects the performance of fully integrated, linear elements subjected to bending loads. These elements function perfectly well under direct or shear loads. Shear locking is not a problem for quadratic elements since their edges are able to curve (see Figure 4–6). The predicted tip displacements for the quadratic elements shown in Table 4–1 are close to the theoretical value. However, quadratic elements will also exhibit some locking if they are distorted or if the bending stress has a gradient, both of which can occur in practical problems.

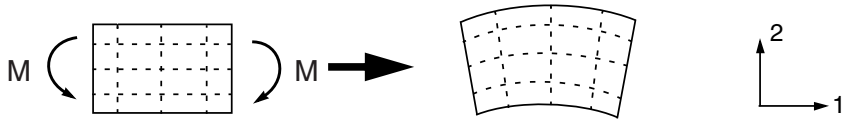


Figure 4–6 Deformation of a fully integrated, quadratic element subjected to bending moment M.

Fully integrated, linear elements should be used only when you are fairly certain that the loads will produce minimal bending in your model. Use a different element type if you have doubts about the type of deformation the loading will create. Fully integrated, quadratic elements can also lock under complex states of stress; thus, you should check the results carefully if they are used exclusively in your model. However, they are very useful for modeling areas where there are local stress concentrations.

Volumetric locking is another form of overconstraint that occurs in fully integrated elements when the material behavior is (almost) incompressible. It causes overly stiff behavior for deformations that should cause no volume changes. It is discussed further in Chapter 10, “Materials.”

4.1.2 Reduced integration

Only quadrilateral and hexahedral elements can use a reduced-integration scheme; all wedge, tetrahedral, and triangular solid elements use full integration, although they can be used in the same mesh with reduced-integration hexahedral or quadrilateral elements.

Reduced-integration elements use one fewer integration point in each direction than the fully integrated elements. Reduced-integration, linear elements have just a single integration point located at the element’s centroid. (Actually, these first-order elements in Abaqus use the more accurate “uniform strain” formulation, where average values of the strain components are computed for the element. This distinction is not important for this discussion.) The locations of the integration points for reduced-integration, quadrilateral elements are shown in Figure 4–7.

Figure 4–7 Integration points in two-dimensional elements with reduced integration.

Abaqus simulations of the cantilever beam problem were performed using the reduced-integration versions of the same four elements utilized previously and using the four finite element meshes shown in Figure 4–3. The results from these simulations are presented in Table 4–2.

Table 4–2 Normalized tip displacements with reduced-integration elements.

Element	Mesh Size (Depth × Length)			
	1 × 6	**2 × 12**	**4 × 12**	**8 × 24**
CPS4R	20.3*	1.308	1.051	1.012
CPS8R	1.000	1.000	1.000	1.000
C3D8R	70.1*	1.323	1.063	1.015
C3D20R	0.999**	1.000	1.000	1.000
* no stiffness to resist the applied load, ** two elements through width				

Linear reduced-integration elements tend to be too flexible because they suffer from their own numerical problem called *hourglassing*. Again, consider a single reduced-integration element modeling a small piece of material subjected to pure bending (see Figure 4–8).

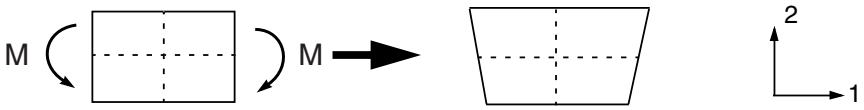


Figure 4–8 Deformation of a linear element with reduced integration subjected to bending moment M.

Neither of the dotted visualization lines has changed in length, and the angle between them is also unchanged, which means that all components of stress at the element's single integration point are zero.

This bending mode of deformation is thus a zero-energy mode because no strain energy is generated by this element distortion. The element is unable to resist this type of deformation since it has no stiffness in this mode. In coarse meshes this zero-energy mode can propagate through the mesh, producing meaningless results.

In Abaqus a small amount of artificial "hourglass stiffness" is introduced in first-order reduced-integration elements to limit the propagation of hourglass modes. This stiffness is more effective at limiting the hourglass modes when more elements are used in the model, which means that linear reduced-integration elements can give acceptable results as long as a reasonably fine mesh is used. The errors seen with the finer meshes of linear reduced-integration elements (see Table 4–2) are within an acceptable range for many applications. The results suggest that at least four elements should be used through the thickness when modeling any structures carrying bending loads with this type of element. When a single linear reduced-integration element is used through the thickness of the beam, all the integration points lie on the neutral axis and the model is unable to resist bending loads. (These cases are marked with a * in Table 4–2.)

Linear reduced-integration elements are very tolerant of distortion; therefore, use a fine mesh of these elements in any simulation where the distortion levels may be very high.

The quadratic reduced-integration elements available in Abaqus/Standard also have hourglass modes. However, the modes are almost impossible to propagate in a normal mesh and are rarely a problem if the mesh is sufficiently fine. The 1×6 mesh of C3D20R elements fails to converge because of hourglassing unless two elements are used through the width, but the more refined meshes do not fail even when only one element is used through the width. Quadratic reduced-integration elements are not susceptible to locking, even when subjected to complicated states of stress. Therefore, these elements are generally the best choice for most general stress/displacement simulations, except in large-displacement simulations involving very large strains and in some types of contact analyses.

4.1.3 Incompatible mode elements

The incompatible mode elements, available primarily in Abaqus/Standard, are an attempt to overcome the problems of shear locking in fully integrated, first-order elements. Since shear locking is caused by the inability of the element's displacement field to model the kinematics associated with bending, additional degrees of freedom, which enhance the element's deformation gradient, are introduced into the first-order element. These enhancements to the deformation gradient allow a first-order element to have a linear variation of the deformation gradient across the element's domain as shown in Figure 4–9(a). The standard element formulation results in a constant deformation gradient across the element as shown in Figure 4–9(b), resulting in the nonzero shear stress associated with shear locking. These enhancements to the deformation gradient are entirely internal to an element and are not associated with nodes positioned along the element edges. Unlike incompatible mode formulations that enhance the displacement field directly, the formulation used in Abaqus does not result in overlapping material or a hole along the boundary between two elements, as shown in Figure 4–10. Furthermore, the formulation used in Abaqus is extended easily to nonlinear, finite-strain simulations, something which is not as easy with the enhanced displacement field elements.

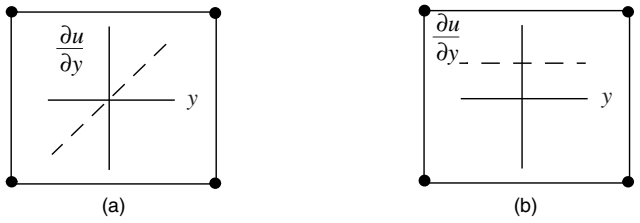


Figure 4–9 Variation of deformation gradient in (a) an incompatible mode (enhanced deformation gradient) element and (b) a first-order element using a standard formulation.

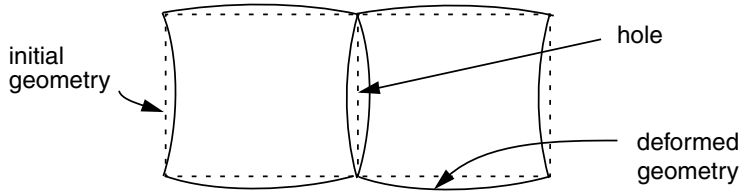


Figure 4–10 Potential kinematic incompatibility between incompatible mode elements that use enhanced displacement fields rather than enhanced deformation gradients. Abaqus uses the latter formulation for its incompatible mode elements.

Incompatible mode elements can produce results in bending problems that are comparable to quadratic elements but at significantly lower computational cost. However, they are sensitive to element distortions. Figure 4–11 shows the cantilever beam modeled with deliberately distorted incompatible mode elements: in one case with "parallel" distortion and in the other with "trapezoidal" distortion.

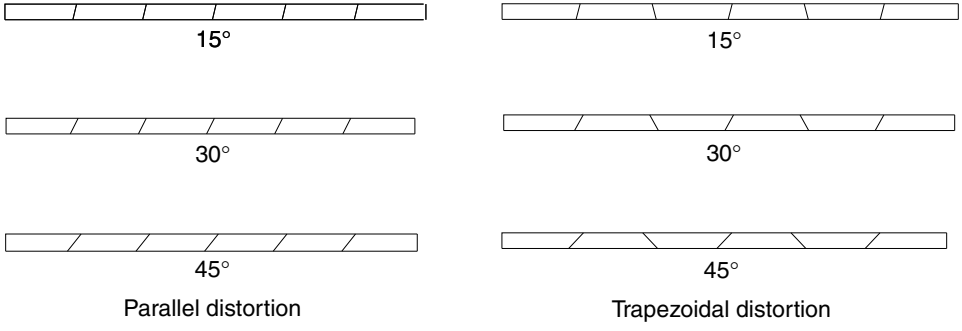


Figure 4–11 Distorted meshes of incompatible mode elements.

Figure 4–12 shows the tip displacements for the cantilever beam models. The tip displacements are normalized with respect to the analytical solution and plotted against the level of element distortion.

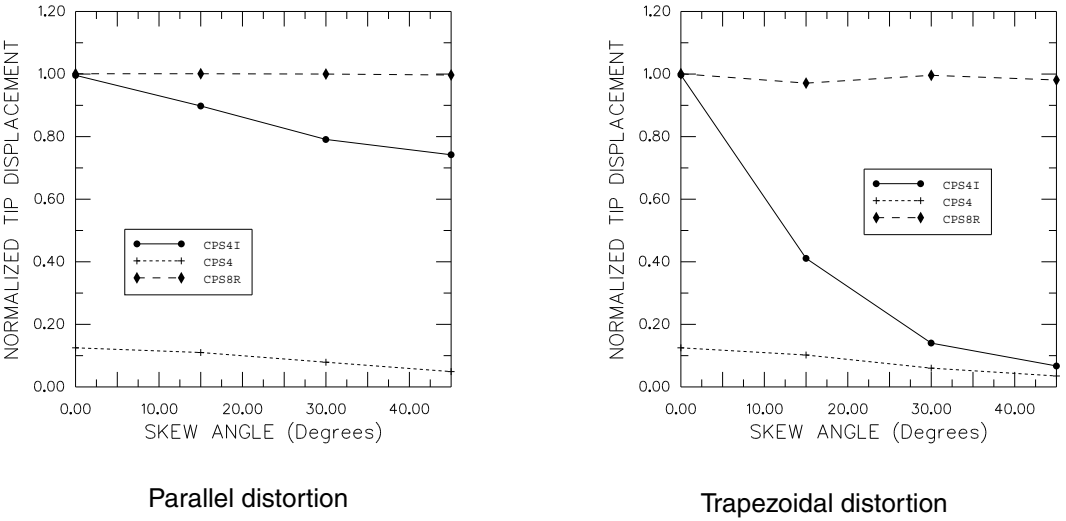


Figure 4–12 Effect of parallel and trapezoidal distortion of incompatible mode elements.

Three types of plane stress elements in Abaqus/Standard are compared: the fully integrated, linear element; the reduced-integration, quadratic element; and the linear, incompatible mode element. The fully integrated, linear elements produce poor results in all cases, as expected. On the other hand, the reduced-integration, quadratic elements give very good results that do not deteriorate until the elements are badly distorted.

When the incompatible mode elements are rectangular, even a mesh with just one element through the thickness of the cantilever gives results that are very close to the theoretical value. However, even quite small levels of trapezoidal distortion make the elements much too stiff. Parallel distortion also reduces the accuracy of the element but to a lesser extent.

Incompatible mode elements are useful because they can provide high accuracy at a low cost if they are used appropriately. However, care must be taken to ensure that the element distortions are small, which may be difficult when meshing complex geometries; therefore, you should again consider using the reduced-integration, quadratic elements in models with such geometries because they show much less sensitivity to mesh distortion. In a severely distorted mesh, however, simply changing the element type will generally not produce accurate results. The mesh distortion should be minimized as much as possible to improve the accuracy of the results.

4.1.4 Hybrid elements

A hybrid element formulation is available for just about every type of continuum element in Abaqus/Standard, including all reduced-integration and incompatible mode elements. Hybrid elements are not available in Abaqus/Explicit. Elements using this formulation have the letter "H" in their names.

Hybrid elements are used when the material behavior is incompressible (Poisson's ratio = 0.5) or very close to incompressible (Poisson's ratio > 0.475). Rubber is an example of a material with incompressible material behavior. An incompressible material response cannot be modeled with regular elements (except in the case of plane stress) because the pressure stress in the element is indeterminate. Consider an element under uniform hydrostatic pressure (Figure 4–13).

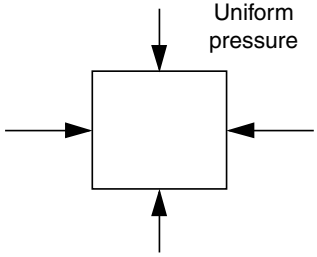


Figure 4–13 Element under hydrostatic pressure.

If the material is incompressible, its volume cannot change under this loading. Therefore, the pressure stress cannot be computed from the displacements of the nodes; and, thus, a pure displacement formulation is inadequate for any element with incompressible material behavior.

Hybrid elements include an additional degree of freedom that determines the pressure stress in the element directly. The nodal displacements are used only to calculate the deviatoric (shear) strains and stresses.

A more detailed description of the analysis of rubber materials is given in Chapter 10, "Materials."

4.2 Selecting continuum elements

The correct choice of element for a particular simulation is vital if accurate results are to be obtained at a reasonable cost. You will undoubtedly develop your own guidelines for selecting elements for your own particular applications as you become more experienced in using Abaqus. However, as you begin to use Abaqus, the guidelines given here may be helpful.

The following recommendations apply to both Abaqus/Standard and Abaqus/Explicit:

- Minimize the mesh distortion as much as possible. Coarse meshes with distorted linear elements can give very poor results.
- Use a fine mesh of linear, reduced-integration elements (CAX4R, CPE4R, CPS4R, C3D8R, etc.) for simulations involving very large mesh distortions (large-strain analysis).
- In three dimensions use hexahedral (brick-shaped) elements wherever possible. They give the best results for the minimum cost. Complex geometries can be difficult to mesh completely with hexahedrons; therefore, wedge and tetrahedral elements may be necessary. The linear versions of these elements, C3D4 and C3D6, are poor elements (fine meshes are needed to obtain accurate results); as a result, these elements should generally be used only when necessary to complete a mesh, and, even then, they should be far from any areas where accurate results are needed.
- Some preprocessors contain free-meshing algorithms that mesh arbitrary geometries with tetrahedral elements. The quadratic tetrahedral elements in Abaqus/Standard (C3D10 or C3D10I) are suitable for general usage; but when used with contact, they should be used only with the "surface-to-surface" contact discretization. An alternative to these elements is the modified quadratic tetrahedral element (C3D10M) available in both analysis products. This element is robust for large-deformation problems and contact problems using either the traditional "node-to-surface" or the "surface-to-surface" contact discretization and exhibits minimal shear and volumetric locking. With either type of element, however, the analysis will take longer to run than an equivalent mesh of hexahedral elements. You should not use a mesh containing only linear tetrahedral elements (C3D4): the results will be inaccurate unless you use an extremely large number of elements.

Abaqus/Standard users should also consider the following recommendations:

- Use quadratic, reduced-integration elements (CAX8R, CPE8R, CPS8R, C3D20R, etc.) for general analysis work, unless you need to model very large strains or have a simulation with complex, changing contact conditions.
- Use quadratic, fully integrated elements (CAX8, CPE8, CPS8, C3D20, etc.) locally where stress concentrations may exist. They provide the best resolution of the stress gradients at the lowest cost.
- For contact problems use a fine mesh of linear, reduced-integration elements or incompatible mode elements (CAX4I, CPE4I, CPS4I, C3D8I, etc.). See Chapter 12, "Contact."

4.3 Example: connecting lug

In this example you will use three-dimensional, continuum elements to model the connecting lug shown in Figure 4–14.

The lug is welded firmly to a massive structure at one end. The other end contains a hole. When it is in service, a bolt will be placed through the hole of the lug. You have been asked to determine the static deflection of the lug when a 30 kN load is applied to the bolt in the negative 2-direction. Because

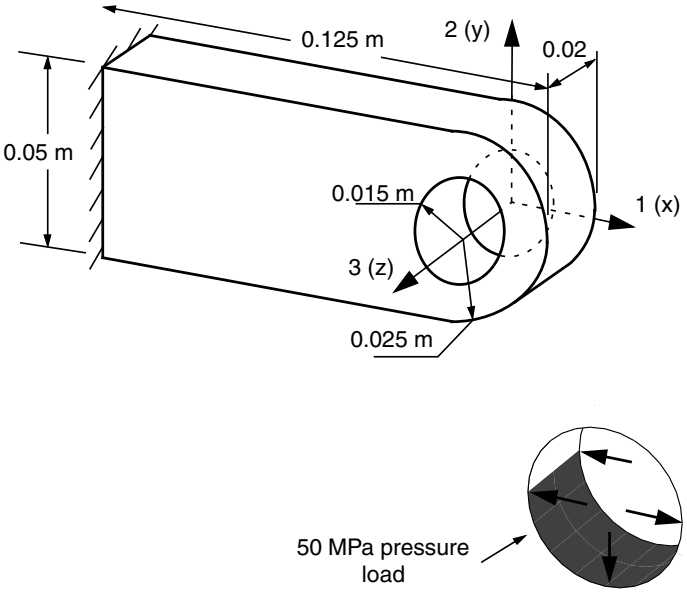


Figure 4–14 Sketch of the connecting lug.

the goal of this analysis is to examine the static response of the lug, you should use Abaqus/Standard as your analysis product. You decide to simplify this problem by making the following assumptions:

- Rather than include the complex bolt-lug interaction in the model, you will use a distributed pressure over the bottom half of the hole to load the connecting lug (see Figure 4–14).
- You will neglect the variation of pressure magnitude around the circumference of the hole and use a uniform pressure.
- The magnitude of the applied uniform pressure will be 50 MPa: 30 kN/ (2 × 0.015 m × 0.02 m).

After examining the static response of the lug, you will modify the model and use Abaqus/Explicit to study the transient dynamic effects resulting from sudden loading of the lug.

4.3.1 Coordinate system

In your model define the global 1-axis to lie along the length of the lug, the global 2-axis to be vertical, and the global 3-axis to lie in the thickness direction. Place the origin of the global coordinate system ($x = 0, y = 0, z = 0$) at the center of the hole on the $z = 0$ face (see Figure 4–14).

4.3.2 Mesh design

You need to consider the type of element that will be used before you start building the mesh for a particular problem. A suitable mesh design that uses quadratic elements may very well be unsuitable if you change to linear, reduced-integration elements. For this example use 20-node hexahedral elements with reduced integration (C3D20R). With the element type selected, you can design the mesh for the connecting lug. The most important decision regarding the mesh design for this application is how many elements to use around the circumference of the lug's hole. A possible mesh for the connecting lug is shown in Figure 4–15; you should build your model to be similar to it.

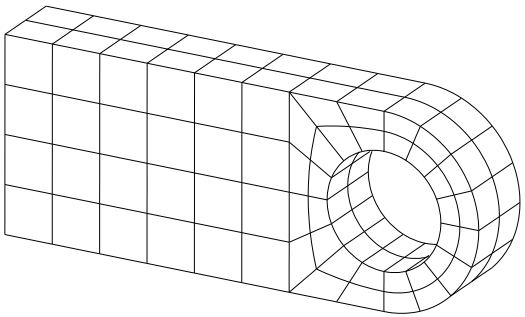


Figure 4–15 Suggested mesh of C3D20R elements for the connecting lug model.

Another thing to consider when designing a mesh is what type of results you want from the simulation. The mesh in Figure 4–15 is rather coarse and, therefore, unlikely to yield accurate stresses. Four quadratic elements per 90° is the minimum number that should be considered for a problem like this one; using twice that many is recommended to obtain reasonably accurate stress results. However, this mesh should be adequate to predict the overall level of deformation in the lug under the applied loads, which is what you were asked to determine. The influence of increasing the mesh density used in this simulation is discussed in "Mesh convergence," Section 4.4.

You need to decide what system of units to use in your model. The SI system of meters, seconds, and kilograms is recommended, but use another system if you prefer.

4.3.3 Preprocessing—creating the model

The model for the overhead hoist in Chapter 2, "Abaqus Basics," was simple enough that the Abaqus input file could be created by typing the input directly into a text editor. This approach clearly is impractical for most real problems; instead, this example and all subsequent examples in the book point

you to the completed input file for the example, and the steps in the examples illustrate the syntax of model and history data in the Abaqus input file. The complete input file for this example, **`lug.inp`**, is available in "Connecting lug," Section A.2.

This example uses the mesh, the node and element sets shown in Figure 4–16, and the pressure load and boundary conditions shown in Figure 4–14.

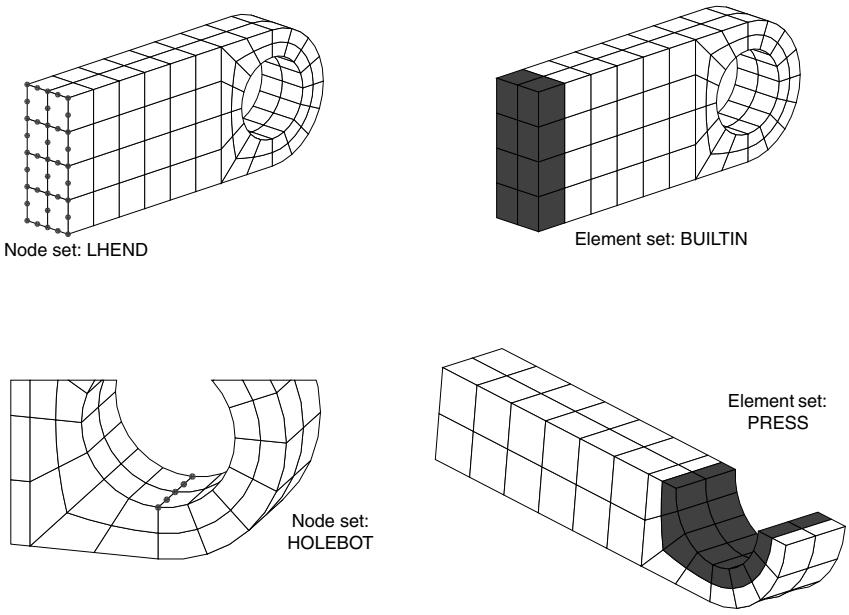
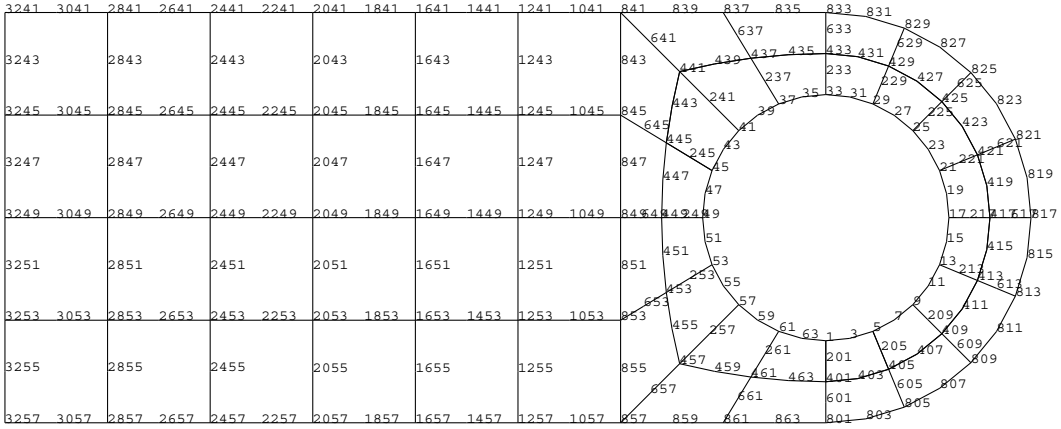


Figure 4–16 Useful node and element sets for the connecting lug simulation.

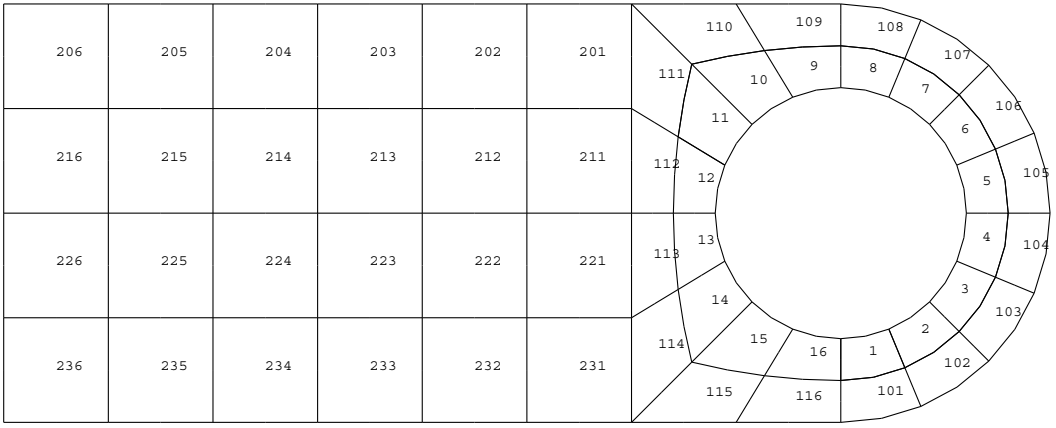
Subsequent steps will add the additional data needed for the model to describe the format of an Abaqus input file. If you would prefer to adjust the mesh and you do not have a preprocessor, use the Abaqus mesh generation options in "Connecting lug," Section A.2. If you wish to create the entire model using Abaqus/CAE, refer to "Example: connecting lug," Section 4.3 of Getting Started with Abaqus: Interactive Edition.

In the description of this simulation that follows, the node and element numbers used are from the model found in "Connecting lug," Section A.2. These node and element numbers are shown in Figure 4–17 and Figure 4–18. If you use a preprocessor, the node and element numbering in your model will almost certainly differ from that shown here. As you make modifications to your input file, remember to use the node and element numbers in your model and not those given here.



Additional planes of nodes in the z-direction are incremented by 5000.

Figure 4–17 Node numbers in the plane $z = 0$.



Additional planes of elements in the z-direction are incremented by 1000.

Figure 4–18 Element numbers in the plane $z = 0$.

4.3.4 Reviewing the input file—the model data

The model data—including the node and element definitions, set definitions, and section and material properties—are discussed in the following sections.

Model description

An Abaqus input file always starts with the *HEADING option. Often the description given in this option by the preprocessor is not very informative, although it might give the date and time when the file was generated. You should provide a suitable title on the data lines of this option so that someone looking at this file can tell what is being modeled and what units you used. The *HEADING option block used in **lug.inp** appears below:

```
*HEADING
Linear Elastic Steel Connecting Lug
S.I. Units (N, kg, m, s)
```

Nodal coordinates and element connectivity

In input files created by a preprocessor, the model's nodal coordinates usually are in one large *NODE option block, with the coordinates specified for each node individually.

The element definitions generated by the preprocessor usually are contained in several *ELEMENT option blocks. Typically, each block contains elements that have the same element section and material properties. In the connecting lug model only one element type is used, and all the elements have the same properties. Therefore, there will probably be a single *ELEMENT option block in your input file. It will look similar to

```
*ELEMENT, TYPE=C3D20R, ELSET=LUG
   1,     1,   401,   405,     5, 10001, 10401, 10405, 10005,
 201,   403,   205,     3, 10201, 10403, 10205, 10003,
5001,  5401,  5405,  5005
   2,     5,   405,   409,     9, 10005, 10405, 10409, 10009,
 205,   407,   209,     7, 10205, 10407, 10209, 10007,
5005,  5405,  5409,  5009
.......
```

Here three data lines are used to define the connectivity of one C3D20R element completely (a minimum of two is required). If a data line in an *ELEMENT option block ends with a comma, it indicates that the next data line contains more nodes defining the current element. The parameter **ELSET=LUG** indicates that all the elements defined in the following data lines will be stored in an element set called **LUG**. If your model does not have a descriptive element set name in the *ELEMENT option, change it to **LUG**.

Node and element sets

The node and element sets are important components of an Abaqus input file because they allow you to assign loads, boundary conditions, and material properties efficiently. They also offer great flexibility in defining the output that your simulation will produce and make it much easier to understand the input file.

Some preprocessors, such as Abaqus/CAE, will allow you to select and name groups of entities, such as nodes and elements, as you build the model; when the Abaqus input file is created, node and element sets are generated from these groups.

You define sets using the *NSET and *ELSET options in the input file. The name of a set is specified with either the NSET or ELSET parameter. The data lines list the nodes or elements that are included in the set. Each data line can contain up to 16 numbers, and there can be as many data lines as required. For example, the node set **LHEND** (see Figure 4–16) can be defined as

```
*NSET, NSET=LHEND
 3241,  3243,  3245,  3247,  3249,  3251,  3253,  3255, 3257,
 8241,  8245,  8249,  8253,  8257, 13241, 13243, 13245, 13247,
13249, 13251, 13253, 13255, 13257, 18241, 18245, 18249, 18253,
18257, 23241, 23243, 23245, 23247, 23249, 23251, 23253, 23255,
23257
```

If you are adding a node or element set to the input file with an editor and the identification numbers follow a regular pattern, the GENERATE parameter allows a range of nodes to be included by specifying the beginning node number, ending node number, and the increment in node numbers. For example, the node set **LHEND** could be defined as follows:

```
*NSET, NSET=LHEND, GENERATE
 3241,  3257, 2
 8241,  8257, 4
13241, 13257, 2
18241, 18257, 4
23241, 23257, 2
```

Sets can also be created by referring to other sets. If the preprocessor that you used did not create the element set **BUILTIN** or the node set **HOLEBOT** that are shown in Figure 4–16, add them to your input file using an editor; they will be essential in limiting the output during the simulation. You should also create the element set **PRESS** shown in Figure 4–16. Remember to use the node and element numbers in your model and not those shown in the figure.

Section properties

Look up the C3D20R element in Chapter 28, “Continuum Elements,” of the Abaqus Analysis User’s Manual to determine the correct element section option and the required data that must be specified for this element. You will discover that the C3D20R element uses the *SOLID SECTION option to define the element’s section properties. Because this is a three-dimensional element, Abaqus needs no additional geometric data for the element section.

Element set **LUG** contains all the elements, so that element set is suitable for this example. The following element section option statement is used for this example:

```
*SOLID SECTION, ELSET=LUG, MATERIAL=STEEL
```

If you did not define an element set with the name **LUG**, use the name of whatever element set contains all the elements in your model as the value of the ELSET parameter. The material property definition in the model will have the name **STEEL**.

Materials

The connecting lug is made of a mild steel and, thus, has isotropic, linear elastic material behavior under the loads being applied. Assume that E = 200 GPa and that ν = 0.3. These are given on the data line of the *ELASTIC option (remember the overhead hoist example in Chapter 2, "Abaqus Basics"). The following material property definition specifies these properties in the input file:

```
*MATERIAL, NAME=STEEL
*ELASTIC
200.E9, 0.3
```

The value for the NAME parameter on the *MATERIAL option must match the value of the MATERIAL parameter on the *SOLID SECTION option.

4.3.5 Reviewing the input file—the history data

The history data portion of the input file starts at the first *STEP option. Many preprocessors create a linear static step in the input file by default. This example will use a general static step. The following options define the step:

```
*STEP
<possibly a title describing this step>
*STATIC
```

If these options are not in your input file, add them at the end of the existing data. It is easier for someone else to understand your model if you use the data lines following the *STEP option to add a suitable title describing the event being simulated in the step.

Boundary conditions

In the model of the connecting lug, all the nodes need to be constrained in all three directions at the left-hand end where it is attached to its parent structure (see Figure 4–19).

In this example, each constrained degree of freedom is specified individually in the *BOUNDARY option block, as shown below:

```
*BOUNDARY
3241, 1,1
3241, 2,2
3241, 3,3
......
```

If a large number of nodes are constrained, these data can occupy a great deal of space in the computer's memory. Where a number of nodes all have the same boundary conditions, it is more

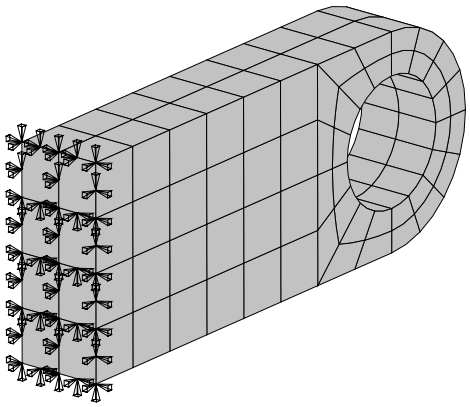


Figure 4–19 Boundary conditions on the connecting lug.

efficient to apply the constraints directly to a node set containing all the nodes. Thus, in the lug model we prefer to create the node set **LHEND** to specify the constraints:

```
*BOUNDARY
LHEND, ENCASTRE
```

If you think that you defined the boundary conditions incorrectly, you can display them in Abaqus/Viewer and compare them with the boundary conditions shown in Figure 4–19. The postprocessing instructions given at the end of "Postprocessing," Section 2.3.9, discuss how to do this.

Loading

The lug carries a pressure of 50 MPa distributed around the bottom-half of the hole. Pressure loads can be defined conveniently using the preprocessor by selecting the element faces to which the load is applied. In the connecting lug input file, these loads will appear as a *DLOAD option block. For example, the *DLOAD option block for the connecting lug may look like

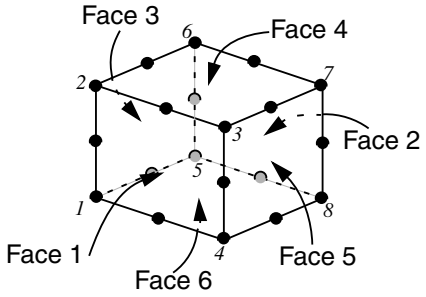
```
*DLOAD
   1,  P6, 5.E+07
   2,  P6, 5.E+07
   3,  P6, 5.E+07
   4,  P6, 5.E+07
  13,  P6, 5.E+07
...

1015,  P6, 5.E+07
1016,  P6, 5.E+07
```

The format of each data line is

<element or element set name>, *<load ID>*, *<load magnitude>*

In this case the "load ID" consists of the letter "P" followed by the number of the element face to which pressure is applied. The face numbers depend on the connectivity of the element and are defined for each element type in the Abaqus Analysis User's Manual. For the three-dimensional hexahedral elements used in this example, the face numbers are shown in Figure 4–20.



Face 1	1-2-3-4 face
Face 2	5-8-7-6 face
Face 3	1-5-6-2 face
Face 4	2-6-7-3 face
Face 5	3-7-8-4 face
Face 6	4-8-5-1 face

Figure 4–20 Face numbers on hexahedral elements.

In the model, as defined in "Connecting lug," Section A.2, the pressure is applied to face 6 of the elements around the bottom of the hole, so the load ID is "P6."

For meshes generated with a preprocessor, the face numbers and, hence, the load IDs will depend on how the mesh is generated. Some preprocessors, such as Abaqus/CAE, can determine the correct load ID automatically; this makes it very easy to specify pressure loads on complicated meshes. However, this method tends to produce long lists of data lines in the input file. In models where the same load ID and load magnitude are used for each element, you can use an element set—which is more efficient—to apply the pressure loads. For example, in this model the *DLOAD option block may appear as

```
*DLOAD
PRESS,  P6, 5.E+07
```

where we have made use of the element set **PRESS** whose members are shown in Figure 4–16.

Output requests

By default, many preprocessors create an Abaqus input file that has a large number of output request options. These requests are in addition to the output database file request that is generated automatically by Abaqus. If, when you edit your input file, you find that these additional output options were created, delete them because they will generally generate too much unnecessary output.

You were asked to determine the deflection of the connecting lug when the load is applied. A simple method for obtaining this result is to print out all the displacements in the model. However, it is likely that the location on the lug with the largest deflection is probably going to be on the bottom of the hole, where the load is applied. Furthermore, only the displacement in the 2-direction (U2) is going to be of interest. You should have created a node set, **HOLEBOT**, containing those nodes. Use that set to limit the requested displacements to just those five nodes at the bottom of the hole and to limit the output to just the vertical displacements.

```
*NODE PRINT, NSET=HOLEBOT
U2,
```

It is good practice to check that the reaction forces at the constraints balance the applied loads. All the reaction forces at a node can be printed by specifying the variable RF. We again use the node set **LHEND** to limit the output to those nodes that are constrained.

```
*NODE PRINT, NSET=LHEND, TOTAL=YES, SUMMARY=NO
RF,
```

You can define several ∗NODE PRINT and ∗EL PRINT options. The parameter TOTALS=YES causes the sum of the reaction forces at all the nodes in the node set to be printed. The SUMMARY=NO parameter prevents the minimum and maximum values in the table from being printed.

The following commands print the stress tensor (variable S) and the Mises stress (variable MISES) for the elements at the constrained end (element set **BUILTIN**):

```
*EL PRINT, ELSET=BUILTIN
S, MISES
```

You will use the NFORC output variable to create and display free body cuts in "Postprocessing—visualizing the results," Section 4.3.8. The following options write the nodal forces due to the element stresses to the output database while also writing the default output:

```
*OUTPUT, FIELD, VARIABLE=PRESELECT
*ELEMENT OUTPUT
NFORC,
*OUTPUT, HISTORY, VARIABLE=PRESELECT
```

The end of a step is indicated with the option

```
*END STEP
```

This input option must be the last option in your model.

4.3.6 Running the analysis

If you modified any input data, store the input in a file called **lug.inp** (an example file is listed in "Connecting lug," Section A.2). Then, run the simulation using the command:

```
abaqus job=lug interactive
```

When the job has completed, check the data file, **lug.dat**, for any errors or warnings. If there are any errors, correct the input file and run the simulation again. If you have problems correcting any errors, try comparing your input file to the one given in "Connecting lug," Section A.2. Check that you have the correct parameters for each input option.

4.3.7 Results

When the job has completed successfully, look at the three tables of output that you requested. They will be found at the end of the data file. A portion of the table of element stresses is shown in Figure 4–21. The maximum Mises stress at the built-in end is approximately 306 MPa.

THE FOLLOWING TABLE IS PRINTED AT THE INTEGRATION POINTS FOR ELEMENT TYPE C3D20R AND ELEMENT SET BUILTIN

ELEMENT	PT	FOOT-NOTE	S11	S22	S33	S12	S13	S23	MISES
206	1		2.8192E+08	-8.1398E+06	-1.3867E+07	-6.9975E+06	-1.1688E+07	1.1556E+06	2.9392E+08
206	2		3.4766E+08	8.7629E+07	8.1158E+07	-4.9896E+07	4.2710E+07	3.1290E+06	2.8690E+08
206	3		1.8341E+08	1.3272E+06	-8.9091E+06	-3.3674E+07	-6.3447E+06	1.7790E+06	1.9661E+08
206	4		1.9471E+08	3.8981E+07	3.8422E+07	-2.4493E+07	2.7244E+07	3.1046E+06	1.6851E+08
206	5		3.0367E+08	-1.1909E+06	-2.7817E+06	-8.2581E+06	-4.0589E+06	-1.8407E+05	3.0608E+08
206	6		3.2968E+08	7.9725E+07	7.4055E+07	-5.6416E+07	9.2002E+06	-7.8331E+04	2.7153E+08
206	7		1.9944E+08	7.8575E+06	-1.0716E+06	-3.4469E+07	-2.3479E+06	5.4628E+05	2.0512E+08
206	8		1.8060E+08	3.3280E+07	3.2765E+07	-3.0944E+07	5.6498E+06	-7.4119E+04	1.5731E+08
.									
.									
1236	1		-1.9946E+08	-7.8863E+06	1.0719E+06	-3.4403E+07	-2.3479E+06	-5.4545E+05	2.0510E+08
1236	2		-1.8062E+08	-3.3293E+07	-3.2771E+07	-3.0908E+07	5.6503E+06	7.4267E+04	1.5730E+08
1236	3		-3.0367E+08	1.1878E+06	2.7818E+06	-8.2327E+06	-4.0583E+06	1.8502E+05	3.0607E+08
1236	4		-3.2963E+08	-7.9705E+07	-7.4039E+07	-5.6389E+07	9.1989E+06	7.8203E+04	2.7148E+08
1236	5		-1.8343E+08	-1.3529E+06	8.9107E+06	-3.3606E+07	-6.3449E+06	-1.7770E+06	1.9658E+08
1236	6		-1.9474E+08	-3.8996E+07	-3.8431E+07	-2.4460E+07	2.7246E+07	-3.1025E+06	1.6851E+08
1236	7		-2.8193E+08	8.1369E+06	1.3864E+07	-6.9711E+06	-1.1686E+07	-1.1543E+06	2.9393E+08
1236	8		-3.4761E+08	-8.7610E+07	-8.1144E+07	-4.9872E+07	4.2703E+07	-3.1275E+06	2.8686E+08
MAXIMUM			3.4766E+08	8.7629E+07	8.1158E+07	-6.9711E+06	4.2710E+07	3.1290E+06	3.0608E+08
ELEMENT			206	206	206	236	206	206	206
MINIMUM			-3.4761E+08	-8.7610E+07	-8.1144E+07	-5.6416E+07	-4.2710E+07	-3.1290E+06	3.5722E+07
ELEMENT			236	236	236	206	1206	1206	226

Integration point at which results are given.

Figure 4–21 Table of element stresses in the data file.

The tables showing the displacements of the nodes along the bottom of the hole and the reaction forces at the constrained nodes are shown in Figure 4–22 and Figure 4–23, respectively.

```
 THE FOLLOWING TABLE IS PRINTED FOR NODES BELONGING TO NODE SET HOLEBOT

  NODE FOOT-   U2
       NOTE

     1      -3.1342E-04
  5001      -3.1348E-04
 10001      -3.1349E-04
 15001      -3.1348E-04
 20001      -3.1342E-04

MAXIMUM     -3.1342E-04
AT NODE      20001

MINIMUM     -3.1349E-04
AT NODE      10001
```

Figure 4–22 Table of nodal displacements in the data file.

The bottom of the hole in the lug has displaced about 0.3 mm. The total reaction force in the 2-direction at the constrained nodes is equal and opposite to the applied load in that direction of −30 kN.

4.3.8 Postprocessing—visualizing the results

Once you have looked at the results in the data file, start Abaqus/Viewer by typing the following command at the operating system prompt:

```
abaqus viewer odb=lug
```

Plotting the deformed shape

From the main menu bar, select **Plot→Deformed Shape**; or use the tool in the toolbox. Figure 4–24 displays the deformed model shape at the end of the analysis. What is the displacement magnification level?

Changing the view

The default view is isometric. You can change the view using the options in the **View** menu or the view tools in the **View Manipulation** toolbar. You can also specify a view by entering values for rotation angles, viewpoint, zoom factor, or fraction of viewport to pan. Direct view manipulation is also available using the 3D compass.

To manipulate the view using the 3D compass:

- Click and drag one of the straight axes of the 3D compass to pan along an axis.
- Click and drag any of the quarter-circular faces on the 3D compass to pan along a plane.
- Click and drag one of the three arcs along the perimeter of the 3D compass to rotate the model about the axis that is perpendicular to the plane containing the arc.
- Click and drag the free rotation handle (the point at the top of the 3D compass) to rotate the model freely about its pivot point.

THE FOLLOWING TABLE IS PRINTED FOR NODES BELONGING TO NODE SET LHEND

NODE	FOOT-NOTE	RF1	RF2	RF3
3241		872.9	765.2	-936.5
3243		-1.0792E+04	-139.6	-2692.
3245		2544.	29.24	-636.7
3247		-3471.	248.1	-879.4
3249		-0.1244	-366.6	9.4686E-02
3251		3473.	247.2	879.7
3253		-2543.	29.40	636.9
3255		1.0792E+04	-140.0	2692.
3257		-873.2	765.1	936.4
8241		-1.2520E+04	3365.	-150.5
8245		-1.1681E+04	1683.	-221.0
8249		0.7131	761.5	-1.5664E-02
8253		1.1682E+04	1681.	220.9
8257		1.2518E+04	3364.	150.4
13241		2632.	1467.	-1.2818E-11
13243		-1.8432E+04	256.3	5.4115E-10
13245		6063.	273.5	-1.1869E-10
13247		-5943.	946.3	1.0687E-10
13249		-0.4196	-470.0	-1.4521E-10
13251		5944.	944.6	-1.1846E-10
13253		-6063.	273.8	-1.7849E-10
13255		1.8431E+04	255.3	-4.3929E-10
13257		-2632.	1467.	-8.8306E-11
18241		-1.2520E+04	3365.	150.5
18245		-1.1681E+04	1683.	221.0
18249		0.7131	761.5	1.5664E-02
18253		1.1682E+04	1681.	-220.9
18257		1.2518E+04	3364.	-150.4
23241		872.9	765.2	936.5
23243		-1.0792E+04	-139.6	2692.
23245		2544.	29.24	636.7
23247		-3471.	248.1	879.4
23249		-0.1244	-366.6	-9.4686E-02
23251		3473.	247.2	-879.7
23253		-2543.	29.40	-636.9
23255		1.0792E+04	-140.0	-2692.
23257		-873.2	765.1	-936.4
TOTAL		1.7806E-09	3.0000E+04	1.6445E-09

The totals of the reaction forces make it easy to check that the sum of the forces acting on the model (applied loads plus the reaction forces) is equal to zero.

Figure 4–23 Table of reaction forces in the data file.

- Click the label for any of the axes on the 3D compass to select a predefined view (the selected axis is perpendicular to the plane of the viewport).
- Double-click anywhere on the 3D compass to specify a view.

Most of the views in this manual are specified directly. This is to allow you to confirm the state of your model by checking against the images in the manual. You are encouraged, however, to practice using the above methods to manipulate your views as you deem fit.

To specify the view:

1. From the main menu bar, select **View→Specify** (or double-click the 3D compass).
   The **Specify View** dialog box appears.

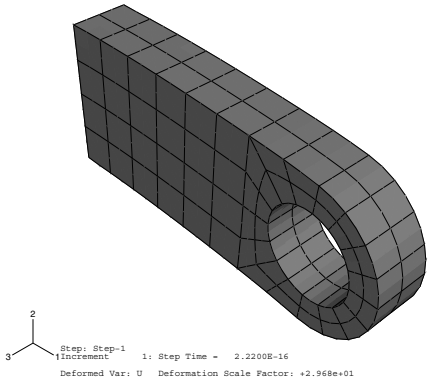


Figure 4–24 Deformed model shape of connecting lug (shaded).

2. From the list of available methods, select **Viewpoint**.

   In the **Viewpoint** method, you enter three values representing the *X*-, *Y*-, and *Z*-position of an observer. You can also specify an up vector. Abaqus positions your model so that this vector points upward.
3. Enter the *X*-, *Y*-, and *Z*-coordinates of the viewpoint vector as `1, 1, 3` and the coordinates of the up vector as `0, 1, 0`.
4. Click **OK**.

   Abaqus/Viewer displays your model in the specified view, as shown in Figure 4–25.

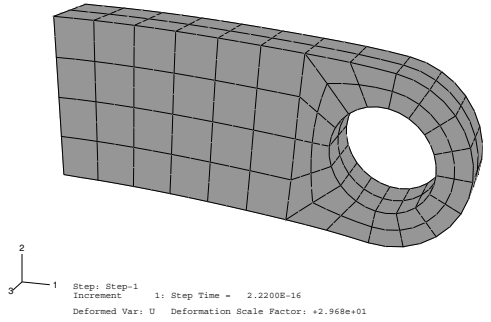


Figure 4–25 Deformed model shape viewed from specified viewpoint.

Visible edges

Several options are available for choosing which edges will be visible in the model display. The previous plots show all exterior edges of the model; Figure 4–26 displays only feature edges.

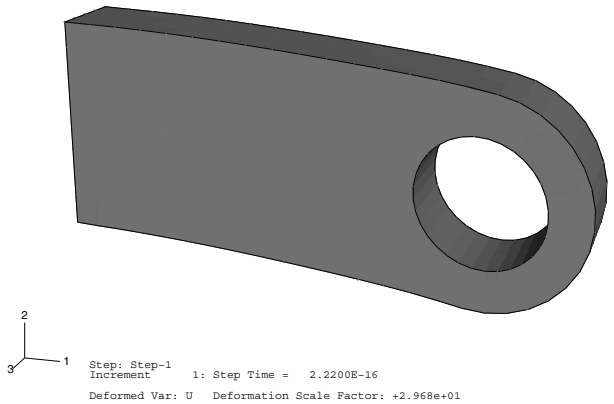


Figure 4–26 Deformed shape with only feature edges visible.

To display only feature edges:

1. From the main menu bar, select **Options→Common**.
   The **Common Plot Options** dialog box appears.
2. Click the **Basic** tab if it is not already selected.
3. From the **Visible Edges** options, choose **Feature edges**.
4. Click **Apply**.
   The deformed plot in the current viewport changes to display only feature edges, as shown in Figure 4–26.

Render style

A shaded plot is a filled plot in which a lightsource appears to be directed at the model. This is the default render style and can be very useful when viewing complex three-dimensional models. Three other render styles provide additional display options: wireframe, hidden line, and filled. You can select a render style from the **Common Plot Options** dialog box or from the tools on the

Render Style toolbar: wireframe, hidden line, filled, and shaded. To display the wireframe plot shown in Figure 4–27, select **Exterior edges** in the **Common Plot Options** dialog box, click **OK** to close the dialog box, and select wireframe plotting by clicking the tool. All subsequent plots will be displayed in the wireframe render style until you select another render style.

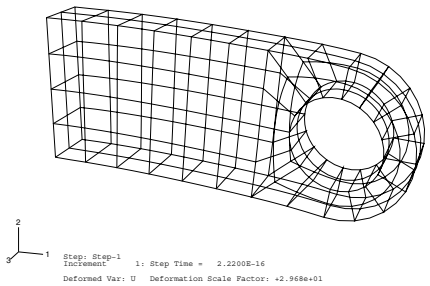


Figure 4–27 Wireframe plot.

A wireframe model showing internal edges can be visually confusing for complex three-dimensional models. You can use the other render style tools to select the hidden line and filled render styles, shown in Figure 4–28 and Figure 4–29, respectively. These render styles are more useful when viewing complex three-dimensional models.

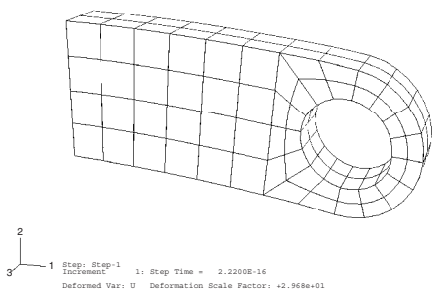


Figure 4–28 Hidden line plot.

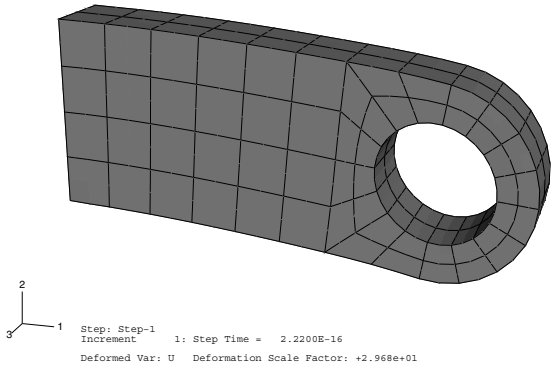


Figure 4–29 Filled element plot.

Contour plots

Contour plots display the variation of a variable across the surface of a model. You can create filled or shaded contour plots of field output results from the output database.

To generate a contour plot of the Mises stress:

1. From the main menu bar, select **Plot→Contours→On Deformed Shape**.

   The filled contour plot shown in Figure 4–30 appears.

   The Mises stress, **`S Mises`**, indicated in the legend title is the default variable chosen by Abaqus for this analysis. You can select a different variable to plot.

2. From the main menu bar, select **Result→Field Output**.

   The **Field Output** dialog box appears; by default, the **Primary Variable** tab is selected.

3. From the list of available output variables, select a new variable to plot.

4. Click **OK**.

   The contour plot in the current viewport changes to reflect your selection.

   Tip: You can also use the **Field Output** toolbar, located above the viewport, to change the displayed field output variable. For more information, see "Using the field output toolbar," Section 42.5.2 of the Abaqus/CAE User's Manual.

Abaqus/Viewer offers many options to customize contour plots. To see the available options, click the **Contour Options** tool in the toolbox. By default, Abaqus/Viewer automatically

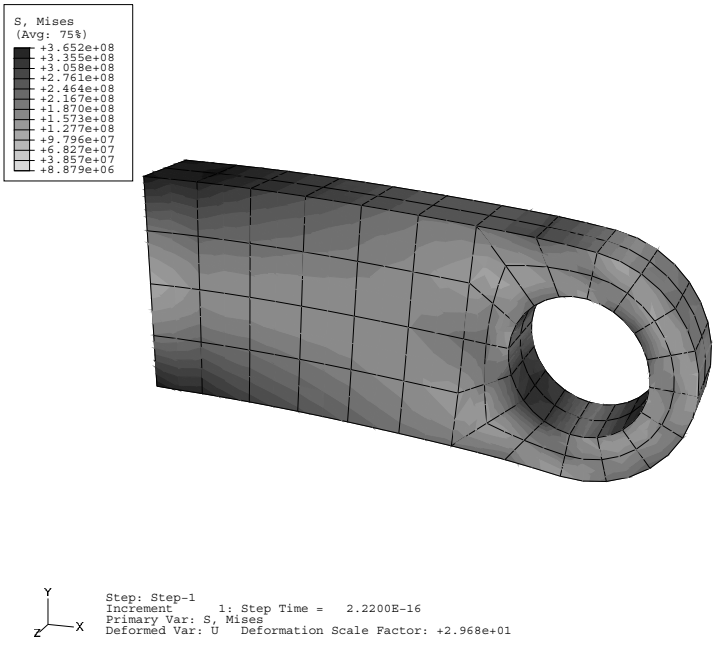


Figure 4–30 Filled contour plot of Mises stress.

computes the minimum and maximum values shown in your contour plots and evenly divides the range between these values into 12 intervals. You can control the minimum and maximum values Abaqus/Viewer displays (for example, to examine variations within a fixed set of bounds), as well as the number of intervals.

To generate a customized contour plot:

1. In the **Basic** tabbed page of the **Contour Plot Options** dialog box, drag the **Contour Intervals** slider to change the number of intervals to nine.
2. In the **Limits** tabbed page of the **Contour Plot Options** dialog box, choose **Specify** beside **Max**; then enter a maximum value of `400E+6`.
3. Choose **Specify** beside **Min**; then enter a minimum value of `60E+6`.
4. Click **OK**.

   Abaqus/Viewer displays your model with the specified contour option settings, as shown in Figure 4–31 (this figure shows Mises stress; your plot will show whichever output variable you have chosen). These settings remain in effect for all subsequent contour plots until you change them or reset them to their default values.

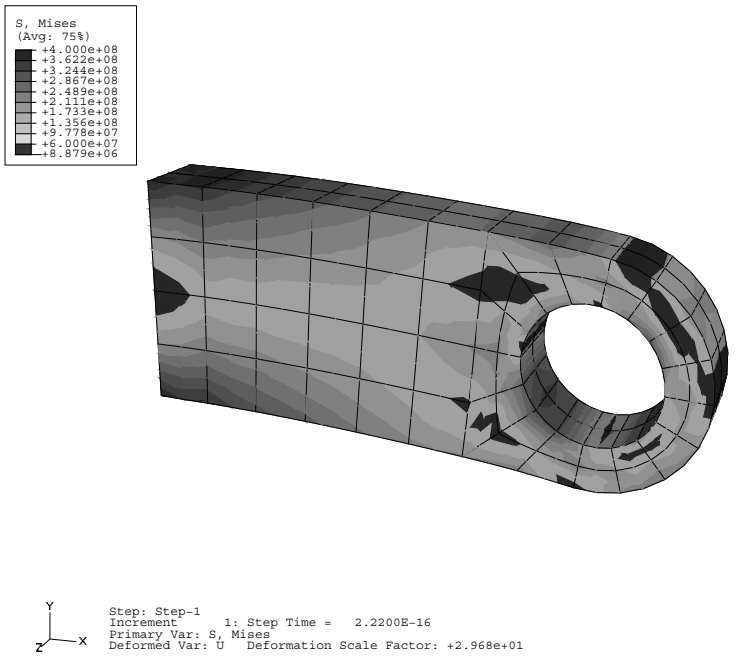


Figure 4–31 Customized plot of Mises stress.

Displaying contour results on interior surfaces

You can cut your model such that interior surfaces are made visible. For example, you may want to examine the stress distribution in the interior of a part. View cuts can be created for such purposes. Here, a simple planar cut is made through the lug to view the Mises stress distribution through the thickness of the part.

To create a view cut:

1. From the main menu bar, select **Tools→View Cut→Create**.
2. In the dialog box that appears, accept the default name and shape. Enter `0,0,0` as the **Origin** of the plane (i.e., a point through which the plane will pass), `1,0,1` as the **Normal axis** to the plane, and `0,1,0` as **Axis 2** of the plane.
3. Click **OK** to close the dialog box and to make the view cut.

   The view appears as shown in Figure 4–32. From the main menu bar, select **Tools→View Cut→Manager** to open the **View Cut Manager**. By default, the regions on and below the cut

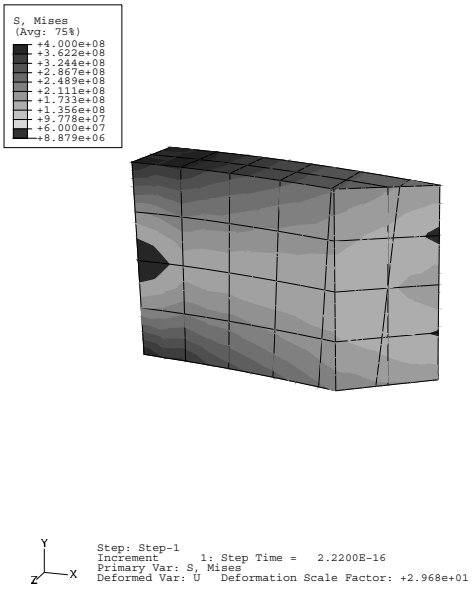


Figure 4–32 Mises stress through the lug thickness.

are displayed (as indicated by the check marks beneath the **on cut** and **below cut** symbols). To translate or rotate the cut, choose **Translate** or **Rotate** from the list of available motions and enter a value or drag the slider at the bottom of the **View Cut Manager**.

4. To view the full model again, toggle off **Cut-4** in the **View Cut Manager**.

   For more information on view cuts, see Chapter 80, "Cutting through a model," of the Abaqus/CAE User's Manual.

Maximum and minimum values

The maximum and minimum values of a variable in a model can be determined easily.

To display the minimum and maximum values of a contour variable:

1. From the main menu bar, select **Viewport→Viewport Annotation Options**; then click the **Legend** tab in the dialog box that appears.

   The **Legend** options become available.

2. Toggle on **Show min/max values**.
3. Click **OK**.

   The contour legend changes to report the minimum and maximum contour values.

One of the goals of this example is to determine the deflection of the lug in the negative 2-direction. You can contour the displacement component of the lug in the 2-direction to determine its peak displacement in the vertical direction as follows. In the **Contour Plot Options** dialog box, click **Defaults** to reset the minimum and maximum contour values and the number of intervals to their default values before proceeding.

To contour the displacement of the connecting lug in the 2-direction:

1. From the list of variable types on the left side of the **Field Output** toolbar, select **Primary** if it is not already selected.

   Tip: You can click on the left side of the **Field Output** toolbar to make your selections from the **Field Output** dialog box instead of the toolbar. If you use the dialog box, you must click **Apply** or **OK** for Abaqus/Viewer to display your selections in the viewport.

2. From the list of available output variables in the center of the toolbar, select output variable **U**.
3. From the list of available components and invariants on the right side of the **Field Output** toolbar, select **U2**.

What is the maximum displacement value in the negative 2-direction?

Displaying a subset of the model

By default, Abaqus/Viewer displays your entire model; however, you can choose to display a subset of your model called a display group. This subset can contain any combination of part instances, geometry (cells, faces, or edges), elements, nodes, and surfaces from the current model or output database. For the connecting lug model you will create a display group consisting of the elements at the bottom of the hole. Since a pressure load was applied to this region, an internal set is created by Abaqus that can be used for visualization purposes.

To display a subset of the model:

1. In the Results Tree, double-click **Display Groups**.

   The **Create Display Group** dialog box opens.

2. From the **Item** list, select **Elements**. From the **Method** list, select **Internal sets**.

   Once you have selected these items, the list on the right-hand side of the **Create Display Group** dialog box shows the available selections.

3. Using this list, identify the set that contains the elements at the bottom of the hole. Toggle on **Highlight items in viewport** below the list so that the outlines of the elements in the selected set are highlighted in red.

4. When the highlighted set corresponds to the group of elements at the bottom of the hole, click **Replace** to replace the current model display with this element set.
   Abaqus/Viewer displays the specified subset of your model.
5. Click **Dismiss** to close the **Create Display Group** dialog box.

When creating an Abaqus model, you may want to determine the face labels for a solid element. For example, you may want to verify that the correct load ID was used when applying pressure loads or when defining surfaces for contact. In such situations you can use the Visualization module to display the mesh after you have run a **datacheck** analysis that creates an output database file.

To display the face identification labels and element numbers on the undeformed model shape:

1. From the main menu bar, select **Options→Common**.
   The **Common Plot Options** dialog box appears.
2. Set the render style to filled; all visible element edges will be displayed for convenience.
   a. Toggle on **Filled** under **Render Style**.
   b. Toggle on **All edges** under **Visible Edges**.
3. Click the **Labels** tab, and toggle on **Show element labels** and **Show face labels**.
4. Click **Apply** to apply the plot options.
5. From the main menu bar, select **Plot→Undeformed Shape**; or use the tool in the toolbox.
   Abaqus/Viewer displays the element and face identification labels in the current display group.
6. Click **Defaults** in the **Common Plot Options** dialog box to restore the default plot settings and then click **OK** to close the dialog box.

Displaying a free body cut

You can define a free body cut to view the resultant forces and moments transmitted across a selected surface of a model. Force vectors are displayed with a single arrowhead and moment vectors with a double arrowhead.

To create a free body cut:

1. To display the entire model in the viewport, select **Tools→Display Group→Plot→All** from the main menu bar.
2. From the main menu bar, select **Tools→Free Body Cut→Manager**.
3. Click **Create** in the **Free Body Cut Manager**.

4. From the dialog box that appears, select **3D element faces** as the **Selection method** and click **Continue**.
5. In the **Free Body Cross-Section** dialog box, select **Surfaces** as the **Item** and **Pick from viewport** as the **Method**.
6. In the prompt area, set the selection method to **by angle** and accept the default angle.
7. Select the surface, highlighted in Figure 4–33, to define the free body cut cross-section.
   a. From the **Selection** toolbar, toggle off the **Select the Entity Closest to the Screen** tool and ensure that the **Select From All Entities** tool is selected.
   b. As you move the cursor in the viewport, Abaqus/CAE highlights all of the potential selections and adds ellipsis marks (...) next to the cursor arrow to indicate an ambiguous selection. Position the cursor so that one of the faces of the desired surface is highlighted, and click to display the first surface selection.

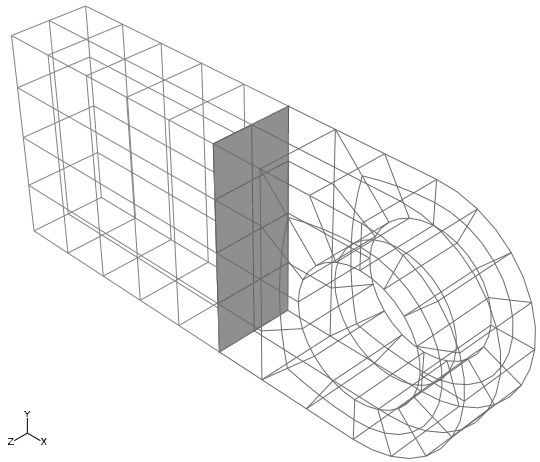


Figure 4–33 Selected faces for the free body cross-section.

   c. Use the **Next** and **Previous** buttons to cycle through the possible selections until the appropriate vertical surface is highlighted, and click **OK**.
8. Click **Done** in the prompt area to indicate your selection is complete. Click **OK** in the **Free Body Cross-Section** dialog box.
9. In the **Edit Free Body Cut** dialog box, accept the default settings for the **Summation Point** and the **Component Resolution**. Click **OK** to close the dialog box.
10. Click **Options** in the **Free Body Cut Manager**.

11. From the **Free Body Plot Options** dialog box, select the **Force** tab in the **Color & Style** tabbed page. Click the resultant color sample to change the color of the resultant force arrow.
12. Once you have selected a new color for the resultant force arrow, click **OK** in the **Free Body Plot Options** dialog box and click **Dismiss** in the **Free Body Cut Manager**.

    The free body cut is displayed in the viewport, as shown in Figure 4–34.

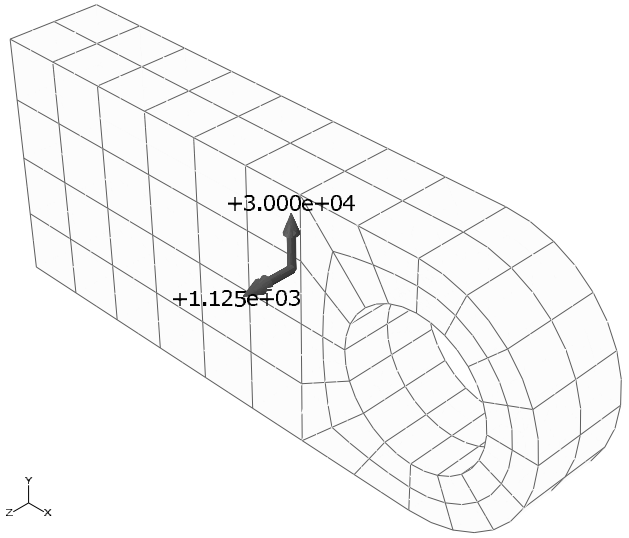


Figure 4–34 Free body cut displayed on the connecting lug.

Generating tabular data reports for subsets of the model

Tabular output data were generated earlier for this model using printed output requests. However, for complicated models it is convenient to write these data for selected regions of the model using Abaqus/Viewer. This is achieved using display groups in conjunction with the report generation feature. For the connecting lug problem we will generate the following tabular data reports:

- Stresses in the elements at the built-in end of the lug (to determine the maximum stress in the lug)
- Reaction forces at the built-in end of the lug (to check that the reaction forces at the constraints balance the applied loads)
- Vertical displacements at the bottom of the hole (to determine the deflection of the lug when the load is applied)

Each of these reports will be generated using display groups whose contents are selected in the viewport. Thus, begin by creating and saving display groups for each region of interest.

To create and save a display group containing the elements at the built-in end:

1. In the Results Tree, double-click **Display Groups**.
2. Choose **Elements** from the **Item** list and **Pick from viewport** as the selection method.
3. Restore the option to select entities closest to the screen.
4. In the prompt area, set the selection method to **by angle**; and click the built-in face of the lug. Click **Done** when all the elements at the built-in face of the lug are highlighted in the viewport. In the **Create Display Group** dialog box, click **Save Selection As**. Save the display group as `built-in elements`.

To create and save a display group containing the nodes at the built-in end:

1. In the **Create Display Group** dialog box, choose **Nodes** from the **Item** list and **Pick from viewport** as the selection method.
2. In the prompt area, set the selection method to **by angle**; and click the built-in face of the lug. Click **Done** when all the nodes on the built-in face of the lug are highlighted in the viewport. In the **Create Display Group** dialog box, click **Save Selection As**. Save the display group as `built-in nodes`.

To create and save a display group containing the nodes at the bottom of the hole:

1. In the **Create Display Group** dialog box, click **Edit Selection** to select a different group of nodes.
2. In the prompt area, set the selection method to **individually**; and select the nodes at the bottom of the hole in the lug, as indicated in Figure 4–35. Click **Done** when all the nodes on the bottom of the hole are highlighted in the viewport. In the **Create Display Group** dialog box, click **Save Selection As**. Save the display group as `nodes at hole bottom`.

   Now generate the reports.

To generate field data reports:

1. In the Results Tree, click mouse button 3 on **built-in elements** underneath the **Display Groups** container. In the menu that appears, select **Plot** to make it the current display group.
2. From the main menu bar, select **Report→Field Output**.
3. In the **Variable** tabbed page of the **Report Field Output** dialog box, accept the default position labeled **Integration Point**. Click the triangle next to **S: Stress components** to expand the list of available variables. From this list, select **Mises** and the six individual stress components: **S11**, **S22**, **S33**, **S12**, **S13**, and **S23**.
4. In the **Setup** tabbed page, name the report `Lug.rpt`. In the **Data** region at the bottom of the page, toggle off **Column totals**.

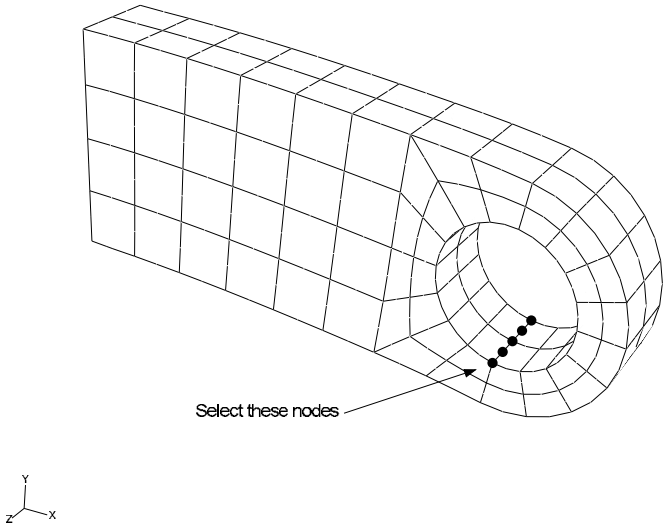


Figure 4–35 Nodes in display group **`nodes at hole bottom`**.

5. Click **Apply**.
6. In the Results Tree, click mouse button 3 on **built-in nodes** underneath the **Display Groups** container. In the menu that appears, select **Plot** to make it the current display group. (To see the nodes, toggle on **Show node symbols** in the **Common Plot Options** dialog box.)
7. In the **Variable** tabbed page of the **Report Field Output** dialog box, change the position to **Unique Nodal**. Toggle off **S: Stress components**, and select **RF1**, **RF2**, and **RF3** from the list of available **RF: Reaction force** variables.
8. In the **Data** region at the bottom of the **Setup** tabbed page, toggle on **Column totals**.
9. Click **Apply**.
10. In the Results Tree, click mouse button 3 on **nodes at hole bottom** underneath the **Display Groups** container. In the menu that appears, select **Plot** to make it the current display group.
11. In the **Variable** tabbed page of the **Report Field Output** dialog box, toggle off **RF: Reaction force**, and select **U2** from the list of available **U: Spatial displacement** variables.
12. In the **Data** region at the bottom of the **Setup** tabbed page, toggle off **Column totals**.
13. Click **OK**.

Open the file **Lug.rpt** in a text editor. A portion of the table of element stresses is shown below. The element data are given at the element integration points. The integration point associated with a given element is noted under the column labeled **Int Pt**. The bottom of the table contains information on the maximum and minimum stress values in this group of elements. The results indicate that the maximum Mises stress at the built-in end is approximately 330 MPa. Your results may differ slightly if your mesh is not identical to the one used here.

```
Field Output Report

Source 1
---------

   ODB: lug.odb
   Step: Step-1
   Frame: Increment      1: Step Time =   2.2200E-16

Loc 1 : Integration point values from source 1

Output sorted by column "Element Label".

Field Output reported at nodes for part: PART-1-1

Element    Int      S.Mises        S.S11        S.S22        S.S33        S.S12
Label      Pt       @Loc 1       @Loc 1       @Loc 1       @Loc 1       @Loc 1
-------------------------------------------------------------------------------
        S.S13        S.S23
       @Loc 1       @Loc 1
--------------------------
  206      1    293.921E+06  281.921E+06  -8.1398E+06 -13.8667E+06 -6.99752E+06
 -11.6881E+06  1.15564E+06
  206      2      286.9E+06  347.661E+06  87.6292E+06  81.1577E+06 -49.8957E+06
  42.7097E+06  3.12903E+06
  206      3    196.605E+06  183.407E+06  1.32717E+06 -8.90914E+06  -33.674E+06
 -6.34469E+06  1.77895E+06
  206      4    168.508E+06  194.713E+06  38.9812E+06  38.4224E+06 -24.4931E+06
  27.2442E+06  3.10456E+06
  206      5    306.077E+06  303.672E+06 -1.19087E+06 -2.78165E+06  -8.2581E+06
 -4.05888E+06  -184.07E+03
  206      6    271.531E+06   329.68E+06  79.7248E+06  74.0551E+06 -56.4163E+06
  9.20019E+06 -78.3313E+03
  206      7    205.123E+06  199.438E+06  7.85751E+06 -1.07157E+06 -34.4693E+06
 -2.34785E+06  546.279E+03
  206      8    157.315E+06  180.601E+06  33.2797E+06  32.7648E+06 -30.9435E+06
  5.64979E+06 -74.1186E+03
    .
    .
  1236     1    205.096E+06 -199.458E+06 -7.88628E+06  1.07185E+06 -34.4032E+06
  -2.3479E+06 -545.449E+03
  1236     2    157.301E+06 -180.618E+06 -33.2934E+06 -32.7715E+06 -30.9083E+06
  5.65027E+06  74.2669E+03
  1236     3    306.071E+06  -303.67E+06  1.18777E+06  2.78175E+06  -8.2327E+06
 -4.05827E+06  185.017E+03
  1236     4     271.48E+06 -329.625E+06 -79.7048E+06 -74.0391E+06 -56.3889E+06
  9.19885E+06  78.2027E+03
  1236     5    196.584E+06 -183.433E+06 -1.35291E+06  8.91071E+06 -33.6059E+06
 -6.34491E+06 -1.77698E+06
  1236     6    168.507E+06 -194.738E+06  -38.996E+06 -38.4311E+06 -24.4598E+06
  27.2461E+06 -3.10252E+06
  1236     7    293.927E+06 -281.931E+06  8.13693E+06  13.8641E+06 -6.97109E+06
 -11.6862E+06 -1.15429E+06
  1236     8    286.857E+06 -347.614E+06 -87.6102E+06 -81.1438E+06 -49.8721E+06
  42.7034E+06 -3.12746E+06


  Minimum      35.7223E+06 -347.614E+06 -87.6102E+06 -81.1438E+06 -56.4163E+06
 -42.7097E+06 -3.12903E+06
```

```
     At Element       226          236          236         1236         1206
        1206         1206
         Int Pt          2            4            4            8            2
           6            6
  Maximum      306.077E+06  347.661E+06  87.6292E+06  81.1577E+06 -6.97109E+06
  42.7097E+06  3.12903E+06
     At Element       206         1206         1206          206         1236
         206          206
         Int Pt          5            6            6            2            7
           2            2
```

How does the maximum value of Mises stress compare to the value reported in the contour plot generated earlier? Do the two maximum values correspond to the same point in the model? The Mises stresses shown in the contour plot have been extrapolated to the nodes, whereas the stresses written to the report file for this problem correspond to the element integration points. Therefore, the location of the maximum Mises stress in the report file is not exactly the same as the location of the maximum Mises stress in the contour plot. This difference can be resolved by requesting that stress output at the nodes (extrapolated from the element integration points and averaged over all elements containing a given node) be written to the report file. If the difference is large enough to be of concern, this is an indication that the mesh may be too coarse.

The table listing the reaction forces at the constrained nodes is shown below. The **`Total`** entry at the bottom of the table contains the net reaction force components for this group of nodes. The results confirm that the total reaction force in the 2-direction at the constrained nodes is equal and opposite to the applied load of −30 kN in that direction.

```
Field Output Report

Source 1
---------

   ODB: lug.odb
   Step: Step-1
   Frame: Increment      1: Step Time =   2.2200E-16

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: PART-1-1

            Node          RF.RF1          RF.RF2          RF.RF3
           Label          @Loc 1          @Loc 1          @Loc 1
-----------------------------------------------------------------
            3241         872.912          765.17        -936.541
            3243    -10.7924E+03        -139.598    -2.69241E+03
            3245     2.5436E+03         29.2367        -636.668
            3247    -3.47143E+03         248.065        -879.401
            3249    -124.431E-03         -366.58     94.6864E-03
     .
     .
           23249    -124.431E-03         -366.58    -94.6864E-03
           23251     3.47251E+03         247.215        -879.699
           23253    -2.54332E+03         29.3956        -636.906
           23255     10.7918E+03        -139.991    -2.69226E+03
           23257        -873.161         765.137        -936.363


  Minimum           -18.4323E+03        -470.038    -2.69241E+03
        At Node          13243           13249            3243
```

```
  Maximum             18.431E+03      3.3654E+03     2.69241E+03
         At Node            13255            8241           23243

           Total     600.502E-06     30.0000E+03    -454.747E-12
```

The table showing the displacements of the nodes along the bottom of the hole (listed below) indicates that the bottom of the hole in the lug has displaced about 0.3 mm.

```
Field Output Report

Source 1
---------

   ODB: lug.odb
   Step: Step-1
   Frame: Increment      1: Step Time =   2.2200E-16

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: PART-1-1

            Node             U.U2
           Label           @Loc 1
---------------------------------
               1    -313.425E-06
           10001    -313.494E-06
           20001    -313.425E-06


  Minimum           -313.494E-06

         At Node           10001
  Maximum           -313.425E-06

         At Node           20001
```

4.3.9 Rerunning the analysis using Abaqus/Explicit

You will now evaluate the dynamic response of the lug when the same load is applied suddenly. Of special interest is the transient response of the lug. You will have to modify the model for the Abaqus/Explicit analysis. Before proceeding, copy the existing input file to a input file named **`lug_xpl.inp`**. Make all subsequent changes to the **`lug_xpl.inp`** input file. Before running the explicit analysis, you will need to change the element type, add a density to the material model, and change the step type. In addition, you should make modifications to the field output requests.

Change element type

Second-order hexahedral elements are not available in the Abaqus/Explicit element library. Thus, you will need to change the element type specified on the ∗ELEMENT option from C3D20R to C3D8R. This change also requires that you edit the element nodal connectivity so that only eight nodes are specified for each element. For example, the following ∗ELEMENT option block was used to define one of the elements in **`lug.inp`**:

```
*ELEMENT, TYPE=C3D20R
   1,     1,   401,   405,     5, 10001, 10401, 10405, 10005,
 201,   403,   205,     3, 10201, 10403, 10205, 10003,
5001,  5401,  5405,  5005
```

In **lug_xpl.inp** this option block has two changes: the element type has been changed to C3D8R and the nodal connectivity consists of the first eight nodes in the original list, which define the corner nodes of the element.

```
*ELEMENT, TYPE=C3D8R
   1,     1,   401,   405,     5, 10001, 10401, 10405, 10005
```

Because the C3D8R element employs reduced integration, use the enhanced strain algorithm to control hourglassing. You can specify enhanced hourglassing with the *SECTION CONTROLS option:

```
*SOLID SECTION, ELSET=LUG, MATERIAL=STEEL, CONTROLS=EC-1
*SECTION CONTROLS, NAME=EC-1, HOURGLASS=ENHANCED
```

Edit the material definition

Since Abaqus/Explicit performs a dynamic analysis, a complete material definition requires that you specify the material density. For this problem assume the density is equal to 7800 kg/m^3.

You can modify the material definition by adding the *DENSITY option to the material option block. The complete material definition for the connecting lug is:

```
*MATERIAL, NAME=STEEL
*ELASTIC
200.E9, 0.3
*DENSITY
7800.,
```

Replace the static step with a dynamic, explicit step

Revise the step definition to examine the dynamic response of the lug over a period of 0.005 s. This change requires that you change the *STEP option block, which appeared as follows for the static analysis:

```
*STEP
Apply uniform pressure to the hole
*STATIC
```

Replace this option block with the following one:

```
*STEP
Dynamic lug loading
*DYNAMIC, EXPLICIT
, 0.005
```

Request field output at evenly spaced intervals and default history output

Write field output at 125 equally spaced intervals and also write the default history output. You can specify output at evenly spaced intervals by appending the NUMBER INTERVAL parameter to the *OUTPUT option block. Replace the existing output requests with the following:

```
*OUTPUT, FIELD, NUMBER INTERVAL=125
*NODE OUTPUT
RF, U
*ELEMENT OUTPUT, DIRECTIONS=YES
S,
*OUTPUT, HISTORY, VARIABLE=PRESELECT
```

Save the changes to the input file called **lug_xpl.inp**. Then run the simulation using the command:

```
abaqus job=lug_xpl
```

4.3.10 Postprocessing the dynamic analysis results

In the static analysis performed with Abaqus/Standard you examined the deformed shape of the lug as well as stress and displacement output. For the Abaqus/Explicit analysis you can similarly examine the deformed shape, stresses, and displacements in the lug. Because transient dynamic effects may result from a sudden loading, you should also examine the time histories for internal and kinetic energy, displacement, and Mises stress.

Open the output database (**.odb**) file created by this job.

Plotting the deformed shape

From the main menu bar, select **Plot→Deformed Shape**; or use the tool in the toolbox. Figure 4–36 displays the deformed model shape at the end of the analysis. As discussed earlier, Abaqus/Explicit assumes large deformation theory by default; thus, the deformation scale factor is automatically set to 1. If the displacements are too small to be seen, scaling can be applied to aid the study of the response.

To see the vibrations in the lug more clearly, change the deformation scale factor to 50. In addition, animate the time history of the deformed shape of the lug and decrease the frame rate of the time history animation.

The time history animation of the deformed shape of the lug shows that the suddenly applied load induces vibrations in the lug. Additional insights about the behavior of the lug under this type

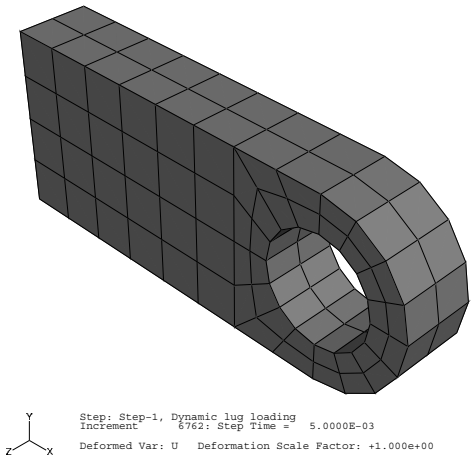


Figure 4–36 Deformed model shape for the explicit analysis (shaded).

of loading can be gained by plotting the kinetic energy, internal energy, displacement, and stress in the lug as a function of time. Some of the questions to consider are:

1. Is energy conserved?
2. Was large-displacement theory necessary for this analysis?
3. Are the peak stresses reasonable? Will the material yield?

X–Y plotting

X–Y plots can display the variation of a variable as a function of time. You can create *X–Y* plots from field and history output.

To create *X–Y* plots of the internal and kinetic energy as a function of time:

1. In the Results Tree, expand the **History Output** container underneath the output database named **`lug_xpl.odb`**.
2. The list of all the variables in the history portion of the output database appears; these are the only history output variables you can plot.

   From the list of available output variables, double-click **ALLIE** to plot the internal energy for the whole model.

Abaqus reads the data for the curve from the output database file and plots the graph shown in Figure 4–37.

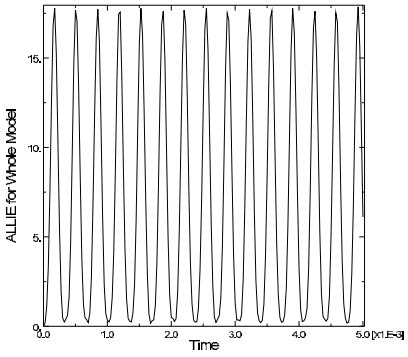


Figure 4–37 Internal energy for the whole model.

3. Repeat this procedure to plot **ALLKE**, the kinetic energy for the whole model (shown in Figure 4–38).

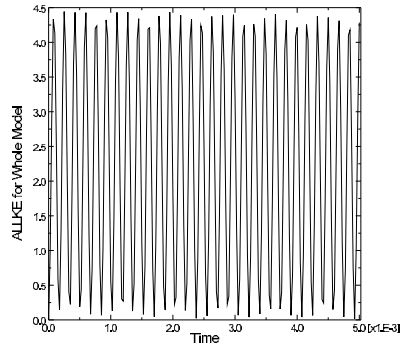


Figure 4–38 Kinetic energy for the whole model.

Both the internal energy and the kinetic energy show oscillations that reflect the vibrations of the lug. Throughout the simulation, kinetic energy is transformed into internal (strain) energy and vice-versa. Since the material is linear elastic, total energy is conserved. This can be seen by plotting **ETOTAL**, the total energy of the system, together with **ALLIE** and **ALLKE**. The value of **ETOTAL** is approximately zero throughout the course of the analysis. Energy balances in dynamic analysis are discussed further in Chapter 9, "Nonlinear Explicit Dynamics."

We will examine the nodal displacements at the bottom of the lug hole to evaluate the significance of geometrically nonlinear effects in this simulation.

To generate a plot of displacement versus time:

1. Plot the deformed shape of the lug. In the Results Tree, double-click **XY Data**.
2. In the **Create XY Data** dialog box that appears, select **ODB field output** as the source and click **Continue**.
3. In the **XY Data from ODB Field Output** dialog box that appears, select **Unique Nodal** as the type of position from which the *X*–*Y* data should be read.
4. Click the arrow next to **U: Spatial displacement** and toggle on **U2** as the displacement variable for the *X*–*Y* data.
5. Select the **Elements/Nodes** tab. Choose **Pick from viewport** as the selection method for identifying the node for which you want *X*–*Y* data.
6. Click **Edit Selection**. In the viewport, select one of the nodes on the bottom of the hole as shown in Figure 4–39 (if necessary, change the render style to facilitate your selection). Click **Done** in the prompt area.
7. Click **Plot** in the **XY Data from ODB Field Output** dialog box to plot the nodal displacement as a function of time.

   The history of the oscillation, as shown in Figure 4–40, indicates that the displacements are small (relative to the structure's dimensions).

   Thus, this problem could have been solved adequately using small-deformation theory. This would have reduced the computational cost of the simulation without significantly affecting the results. Nonlinear geometric effects are discussed further in Chapter 8, "Nonlinearity."

We are also interested in the stress history of the connecting lug. The area of the lug near the built-in end is of particular interest because the peak stresses expected to occur there may cause yielding in the material.

To generate a plot of Mises stress versus time:

1. Plot the deformed shape of the lug again.

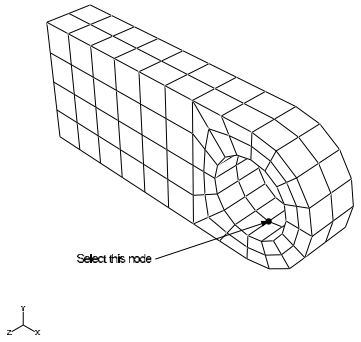


Figure 4–39 Selected node at the bottom of the hole.

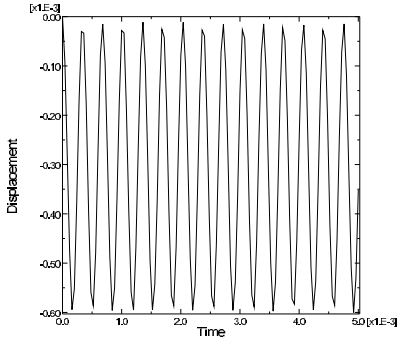


Figure 4–40 Displacement of a node at the bottom of the hole.

2. Select the **Variables** tab in the **XY Data from ODB Field Output** dialog box. Deselect **U2** as the variable for the *X–Y* data plot.
3. Change the **Position** field to **Integration Point**.
4. Click the arrow next to **S: Stress components** and toggle on **Mises** as the stress variable for the *X–Y* data.
5. Select the **Elements/Nodes** tab. Choose **Pick from viewport** as the selection method for identifying the element for which you want *X–Y* data.

6. Click **Edit Selection**. In the viewport, select one of the elements near the built-in end of the lug as shown in Figure 4–41. Click **Done** in the prompt area.

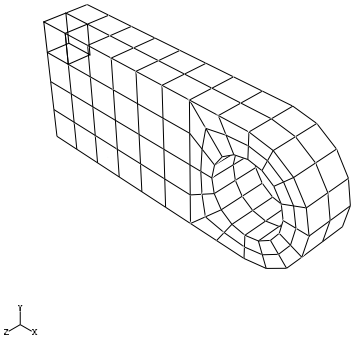


Figure 4–41 Selected element near the built-in end of the lug (hidden).

7. Click **Plot** in the **XY Data from ODB Field Output** dialog box to plot the Mises stress at the selected element as a function of time.

The peak Mises stress is on the order of 550 MPa, as shown in Figure 4–42. This value is larger than the typical yield strength of steel. Thus, the material would have yielded before experiencing such a large stress. Material nonlinearity is discussed further in Chapter 10, "Materials."

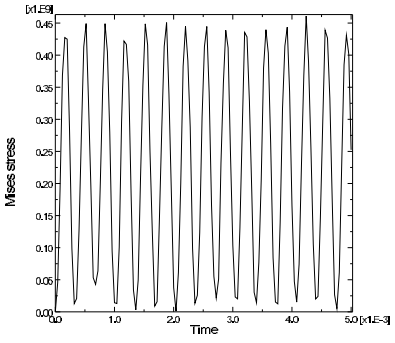


Figure 4–42 Mises stress near the built-in end of the lug.

4.4 Mesh convergence

It is important that you use a sufficiently refined mesh to ensure that the results from your Abaqus simulation are adequate. Coarse meshes can yield inaccurate results in analyses using implicit or explicit methods. The numerical solution provided by your model will tend toward a unique value as you increase the mesh density. The computer resources required to run your simulation also increase as the mesh is refined. The mesh is said to be converged when further mesh refinement produces a negligible change in the solution.

As you gain experience, you will learn to judge what level of refinement produces a suitable mesh to give acceptable results for most simulations. However, it is always good practice to perform a mesh convergence study, where you simulate the same problem with a finer mesh and compare the results. You can have confidence that your model is producing a mathematically accurate solution if the two meshes give essentially the same result.

Mesh convergence is an important consideration in both Abaqus/Standard and Abaqus/Explicit. The connecting lug will be used as an example of a mesh refinement study by further analyzing the connecting lug in Abaqus/Standard using four different mesh densities (Figure 4–43). The number of elements used in each mesh is indicated in the figure.

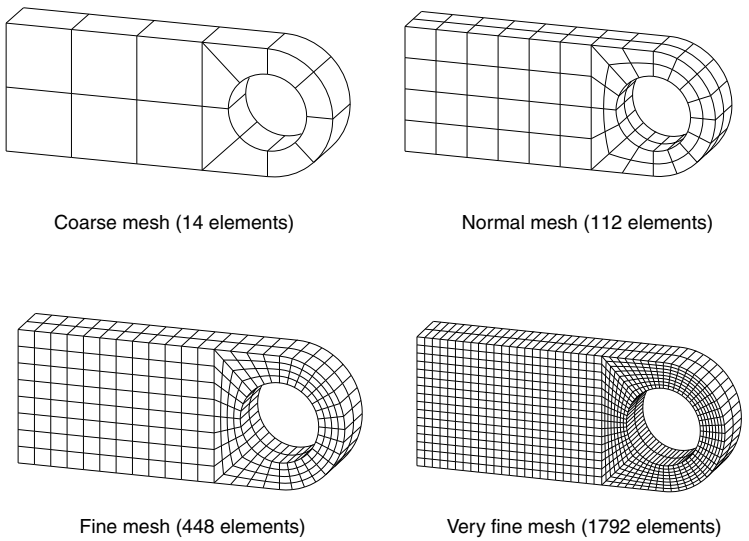


Figure 4–43 Different meshes for the connecting lug problem.

We consider the influence of the mesh density on three particular results from this model:

- The displacement of the bottom of the hole.
- The peak Mises stress at the stress concentration on the bottom surface of the hole.
- The peak Mises stress where the lug is attached to the parent structure.

The locations where the results are compared are shown in Figure 4–44.

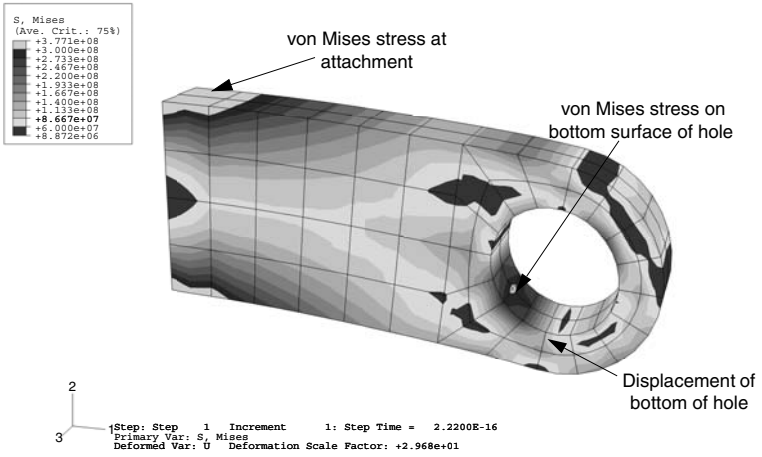


Figure 4–44 Locations where results are compared in the mesh refinement study.

The results for each of the four mesh densities are compared in Table 4–3, along with the CPU time required to run each simulation.

Table 4–3 Results of mesh refinement study.

Mesh	Displacement of bottom of hole	Stress at bottom of hole	Stress at attachment	Relative CPU time
Coarse	3.07E−4	256.E6	312.E6	0.83
Normal	3.13E−4	311.E6	365.E6	1.0
Fine	3.14E−4	332.E6	426.E6	3.2
Very fine	3.15E−4	345.E6	496.E6	13.3

The coarse mesh predicts less accurate displacements at the bottom of hole, but the normal, fine, and very fine meshes all predict similar results. The normal mesh is, therefore, converged as far as the displacements are concerned. The convergence of the results is plotted in Figure 4–45.

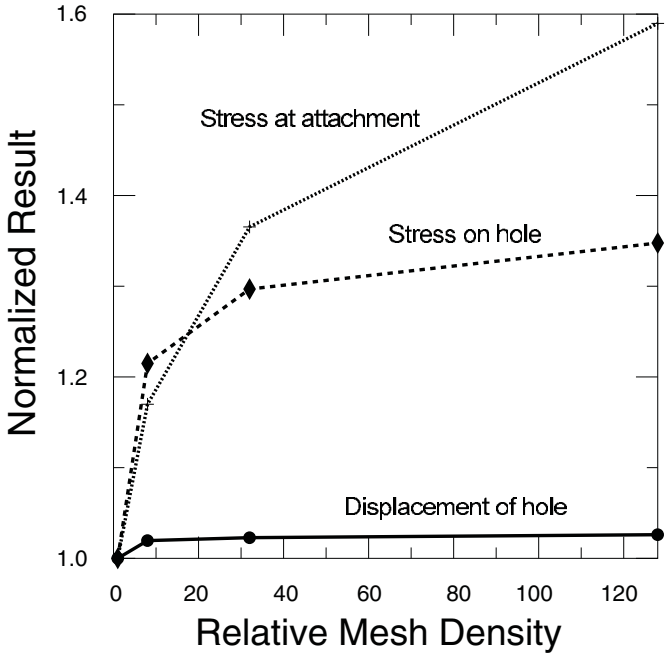


Figure 4–45 Convergence of results in mesh refinement study.

All the results are normalized with respect to the values predicted by the coarse mesh. The peak stress on the bottom of the hole converges much more slowly than the displacements because stress and strain are calculated from the displacement gradients; thus, a much finer mesh is required to predict accurate displacement gradients than is needed to calculate accurate displacements.

Mesh refinement significantly changes the stress calculated at the attachment of the connecting lug; it continues to increase with continued mesh refinement. A stress singularity exists at the corner of the lug where it attaches to the parent structure. Theoretically the stress is infinite at this location; therefore, increasing the mesh density will not produce a converged stress value at this location. This singularity occurs because of the idealizations used in the finite element model. The connection between the lug and the parent structure has been modeled as a sharp corner, and the parent structure has been modeled as rigid. These idealizations lead to the stress singularity. In reality there probably will be a small fillet between the lug and the parent structure, and the parent structure will be deformable, not rigid. If the exact stress in this location is required, the fillet between the components must be modeled accurately (see Figure 4–46) and the stiffness of the parent structure must also be considered.

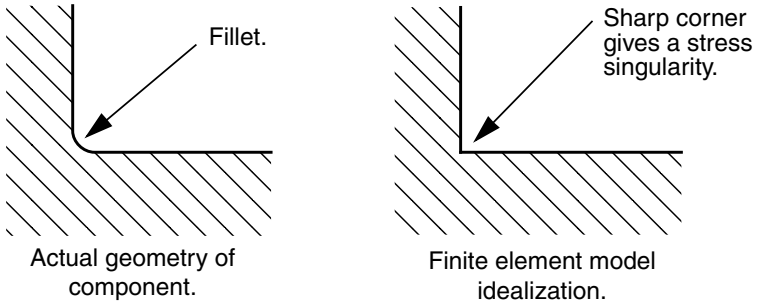


Figure 4–46 Idealizing a fillet as a sharp corner.

It is common to omit small details like fillet radii from a finite element model to simplify the analysis and to keep the model size reasonable. However, the introduction of any sharp corner into a model will lead to a stress singularity at that location. This normally has a negligible effect on the overall response of the model, but the predicted stresses close to the singularity will be inaccurate.

For complex, three-dimensional simulations the available computer resources often dictate a practical limit on the mesh density that you can use. In this case you must use the results from the analysis carefully. Coarse meshes are often adequate to predict trends and to compare how different concepts behave relative to each other. However, you should use the actual magnitudes of displacement and stress calculated with a coarse mesh with caution.

It is rarely necessary to use a uniformly fine mesh throughout the structure being analyzed. You should use a fine mesh mainly in the areas of high stress gradients and use a coarser mesh in areas of low stress gradients or where the magnitude of the stresses is not of interest. For example, Figure 4–47 shows a mesh that is designed to give an accurate prediction of the stress concentration at the bottom of the hole.

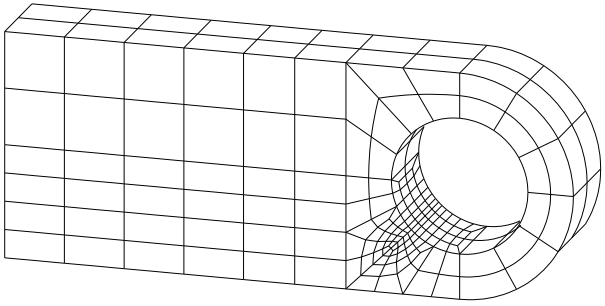


Figure 4–47 Mesh refined around the hole.

A fine mesh is used only in the region of high stress gradients, and a coarse mesh is used elsewhere. The results from an Abaqus/Standard simulation with this locally refined mesh are shown in Table 4–4. This table shows that the results are comparable to those from the very fine mesh, but the simulation with the locally refined mesh required considerably less CPU time than the analysis with the very fine mesh.

Table 4–4 Comparison of very fine and locally refined meshes.

Mesh	Displacement of bottom of hole	Stress at bottom of hole	Relative CPU time
Very fine	3.15E−4	345.E6	22.5
Locally refined	3.14E−4	346.E6	3.44

You can often predict the locations of the highly stressed regions of a model—and, hence, the regions where a fine mesh is required—using your knowledge of similar components or with hand calculations. This information can also be gained by using a coarse mesh initially to identify the regions of high stress and then refining the mesh in these regions. The latter procedure is carried out easily using preprocessors like Abaqus/CAE where the complete numerical model (i.e., material properties, boundary conditions, loads, etc.) can be defined based on the geometry of the structure. It is simple to mesh the geometry coarsely for the initial simulation and then to refine the mesh in the appropriate regions, as indicated by the results from the coarse simulation.

Abaqus provides an advanced feature, called submodeling, that allows you to obtain more detailed (and accurate) results in a region of interest in the structure. The solution from a coarse mesh of the entire structure is used to "drive" a detailed local analysis that uses a fine mesh in this region of interest. (This topic is beyond the scope of this guide. See "Submodeling: overview," Section 10.2.1 of the Abaqus Analysis User's Manual, for further details.)

4.5 Related Abaqus examples

If you are interested in learning more about using continuum elements in Abaqus, you should examine the following problems:

- "Geometrically nonlinear analysis of a cantilever beam," Section 2.1.2 of the Abaqus Benchmarks Manual
- "Spherical cavity in an infinite medium," Section 2.2.4 of the Abaqus Benchmarks Manual
- "Performance of continuum and shell elements for linear analysis of bending problems," Section 2.3.5 of the Abaqus Benchmarks Manual

4.6 Suggested reading

The volume of literature that has been written on the finite element method and the applications of finite element analysis is enormous. In most of the remaining chapters of this guide, a list of suggested books and articles is provided so that you can explore the topics in more depth if you wish. While the advanced references will not be of interest to most users, they provide detailed theoretical information for the interested user.

General texts on the finite element method

- NAFEMS Ltd., *A Finite Element Primer*, 1986.
- Becker, E. B., G. F. Carey, and J. T. Oden, *Finite Elements: An Introduction,* Prentice-Hall, 1981.
- Carey, G. F., and J. T. Oden, *Finite Elements: A Second Course,* Prentice-Hall, 1983.
- Cook, R. D., D. S. Malkus, and M. E. Plesha, *Concepts and Applications of Finite Element Analysis,* John Wiley & Sons, 1989.
- Hughes, T. J. R., *The Finite Element Method*, Prentice-Hall, Inc., 1987.
- Zienkiewicz, O. C., and R. L. Taylor, *The Finite Element Method: Volumes I, II, and III*, Butterworth-Heinemann, 2000.

Performance of linear solid elements

- Prathap, G., "The Poor Bending Response of the Four-Node Plane Stress Quadrilaterals," International Journal for Numerical Methods in Engineering, vol. 21, 825–835, 1985.

Hourglass control in solid elements

- Belytschko, T., W. K. Liu, and J. M. Kennedy, "Hourglass Control in Linear and Nonlinear Problems," Computer Methods in Applied Mechanics and Engineering, vol. 43, 251–276, 1984.
- Flanagan, D. P., and T. Belytschko, "A Uniform Strain Hexahedron and Quadrilateral with Hourglass Control," International Journal for Numerical Methods in Engineering, vol. 17, 679–706, 1981.
- Puso, M. A., "A Highly Efficient Enhanced Assumed Strain Physically Stabilized Hexahedral Element," International Journal for Numerical Methods in Engineering, vol. 49, 1029–1064, 2000.

Incompatible mode elements

- Simo, J. C. and M. S. Rifai, "A Class of Assumed Strain Methods and the Method of Incompatible Modes," International Journal for Numerical Methods in Engineering, vol. 29, 1595–1638, 1990.

4.7 Summary

- The formulation and order of integration used in a continuum element can have a significant effect on the accuracy and cost of the analysis.
- First-order (linear) elements using full integration are prone to shear locking and normally should not be used.
- First-order, reduced-integration elements are prone to hourglassing; sufficient mesh refinement minimizes this problem.
- When using first-order, reduced-integration elements in a simulation where bending deformation will occur, use at least four elements through the thickness.
- Hourglassing is rarely a problem in the second-order, reduced-integration elements in Abaqus/Standard. You should consider using these elements for most general applications when there is no contact.
- The accuracy of the incompatible mode elements available in Abaqus/Standard is strongly influenced by the amount of element distortion.
- The numerical accuracy of the results depends on the mesh that has been used. Ideally a mesh refinement study should be carried out to ensure that the mesh provides a unique solution to the problem. However, remember that using a converged mesh does not ensure that the results from the finite element simulation will match the actual behavior of the physical problem: that also depends on other approximations and idealizations in the model.
- In general, refine the mesh mainly in regions where you want accurate results; a finer mesh is required to predict accurate stresses than is needed to calculate accurate displacements.
- Advanced features such as submodeling are available in Abaqus to help you to obtain useful results for complex simulations.

5. Using Shell Elements

Use shell elements to model structures in which one dimension (the thickness) is significantly smaller than the other dimensions and in which the stresses in the thickness direction are negligible. A structure, such as a pressure vessel, whose thickness is less than 1/10 of a typical global structural dimension generally can be modeled with shell elements. The following are examples of typical global dimensions:

- the distance between supports,
- the distance between stiffeners or large changes in section thickness,
- the radius of curvature, and
- the wavelength of the highest vibration mode of interest.

Abaqus shell elements assume that plane sections perpendicular to the shell midsurface remain plane. Do not be confused into thinking that the thickness must be less than 1/10 of the *element* dimensions. A highly refined mesh may contain shell elements whose thickness is greater than their in-plane dimensions, although this is not generally recommended—continuum elements may be more suitable in such a case.

5.1 Element geometry

Two types of shell elements are available in Abaqus: conventional shell elements and continuum shell elements. Conventional shell elements discretize a reference surface by defining the element's planar dimensions, its surface normal, and its initial curvature. The nodes of a conventional shell element, however, do not define the shell thickness; the thickness is defined through section properties. Continuum shell elements, on the other hand, resemble three-dimensional solid elements in that they discretize an entire three-dimensional body yet are formulated so that their kinematic and constitutive behavior is similar to conventional shell elements. Continuum shell elements are more accurate in contact modeling than conventional shell elements, since they employ two-sided contact taking into account changes in thickness. For thin shell applications, however, conventional shell elements provide superior performance.

In this manual only conventional shell elements are discussed. Henceforth, we will refer to them simply as "shell elements." For more information on continuum shell elements, see "Shell elements: overview," Section 29.6.1 of the Abaqus Analysis User's Manual.

5.1.1 Shell thickness and section points

The shell thickness is required to describe the shell cross-section and must be specified. In addition to specifying the shell thickness, you can choose to have the stiffness of the cross-section calculated during

the analysis or once at the beginning of the analysis. You define the shell thickness using either the *SHELL SECTION or *SHELL GENERAL SECTION option.

If you use the *SHELL SECTION option, Abaqus uses numerical integration to calculate the stresses and strains independently at each section point (integration point) through the thickness of the shell, thus allowing nonlinear material behavior. For example, an elastic-plastic shell may yield at the outer section points while remaining elastic at the inner section points. The location of the single integration point in an S4R (4-node, reduced integration) element and the configuration of the section points through the shell thickness are shown in Figure 5–1.

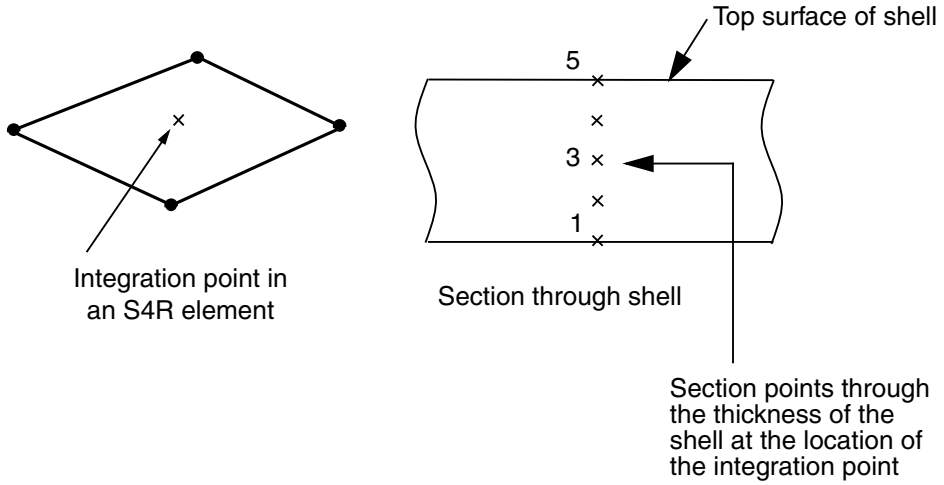


Figure 5–1 Configuration of section points in a numerically integrated shell.

You can specify any odd number of section points through the shell thickness with the *SHELL SECTION option. By default, Abaqus uses five section points through the thickness of a homogeneous shell, which is sufficient for most nonlinear design problems. However, you should use more section points in some complicated simulations, especially when you anticipate reversed plastic bending (nine is normally sufficient in this case). For linear problems three section points provide exact integration through the thickness. However, the *SHELL GENERAL SECTION option is more efficient for linear elastic shells.

If you use the *SHELL GENERAL SECTION option, the material behavior must be linear elastic, as the stiffness of the cross-section is calculated only once at the beginning of the simulation. In this case, all calculations are done in terms of the resultant forces and moments across the entire cross-section. If you request stress or strain output, Abaqus provides default output for the bottom surface, the midplane, and the top surface.

5.1.2 Shell normals and shell surfaces

The connectivity of the shell element defines the direction of the positive normal, as shown in Figure 5–2.

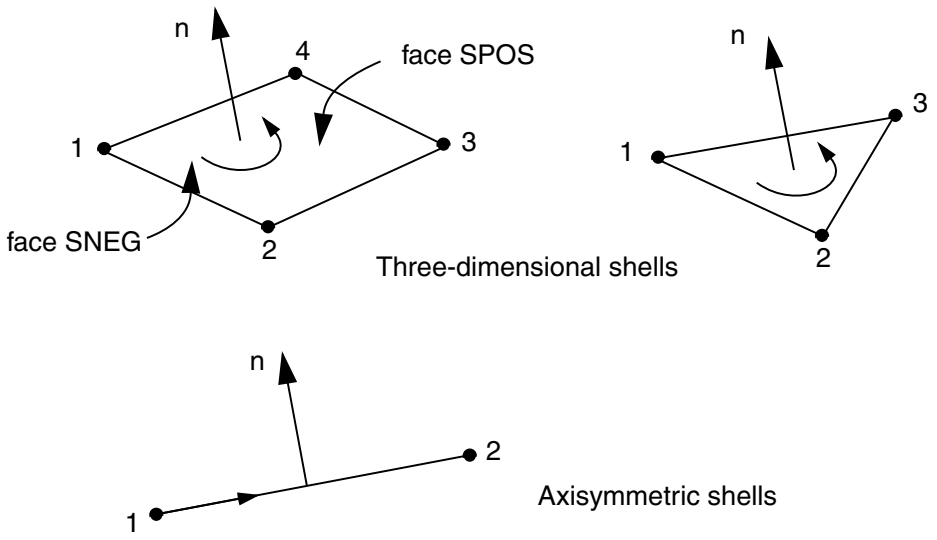


Figure 5–2 Positive normals for shells.

For axisymmetric shell elements the positive normal direction is defined by a 90° counterclockwise rotation from the direction going from node 1 to node 2. For three-dimensional shell elements the positive normal is given by the right-hand rule going around the nodes in the order in which they appear in the element definition.

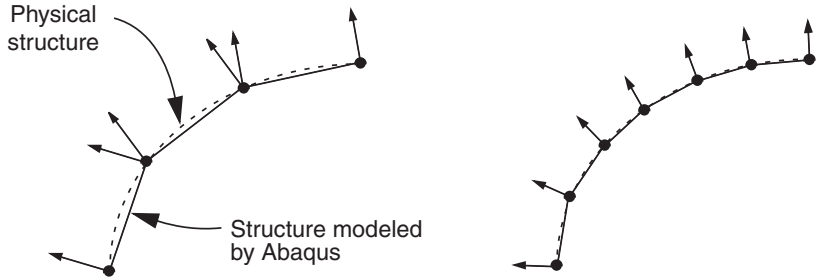
The "top" surface of a shell is the surface in the positive normal direction and is called the SPOS face for contact definition. The "bottom" surface is in the negative direction along the normal and is called the SNEG face for contact definition. Normals should be consistent among adjoining shell elements.

The positive normal direction defines the convention for element-based pressure load application and output of quantities that vary through the shell thickness. A positive element-based pressure load applied to a shell element produces a load that acts in the direction of the positive normal. (The element-based pressure load convention for shell elements is opposite to that for continuum elements; the surface-based pressure load conventions for shell and continuum elements are identical. For more on the difference between element-based and surface-based distributed loads, see "Distributed loads," Section 33.4.3 of the Abaqus Analysis User's Manual.)

5.1.3 Initial shell curvature

Shells in Abaqus (with the exception of element types S3/S3R, S3RS, S4R, S4RS, S4RSW, and STRI3) are formulated as true curved shell elements; true curved shell elements require special attention to accurate calculation of the initial surface curvature. Abaqus automatically calculates the surface normals at the nodes of every shell element to estimate the initial curvature of the shell. The surface normal at each node is determined using a fairly elaborate algorithm, which is discussed in detail in "Defining the initial geometry of conventional shell elements," Section 29.6.3 of the Abaqus Analysis User's Manual.

With a coarse mesh as shown in Figure 5–3, Abaqus may determine several independent surface normals at the same node for adjoining elements. Physically, multiple normals at a single node mean that there is a fold line between the elements sharing the node. While it is possible that you intend to model such a structure, it is more likely that you intend to model a smoothly curved shell; Abaqus will try to smooth the shell by creating an averaged normal at a node.



Coarse mesh

The angle between successive element normals is greater than 20°, so separate normals are retained at each node for adjacent elements, and the behavior is that of a folded sheet.

Refined mesh

There is a single normal at each node for adjacent elements, and the behavior is that of a curved shell.

Figure 5–3 Effect of mesh refinement on the nodal surface normals.

The basic smoothing algorithm used is as follows: if the normals at a node for each shell element attached to the node are within 20° of each other, the normals will be averaged. The averaged normal will be used at that node for all elements attached to the node. If Abaqus cannot smooth the shell, a warning message is issued in the data (`.dat`) file.

You may override the default algorithm. To introduce fold lines into a curved shell or to model a curved shell with a coarse mesh, use the ∗NODE and ∗NORMAL options to define the normals manually. With the ∗NODE option you specify the surface normal at a node as the 4th, 5th, and 6th values on the data line, following the nodal coordinates. A normal you define with ∗NODE is the normal used for

all elements sharing that node, unless *NORMAL is also used. Use the *NORMAL option to specify a normal at a node for selected elements only. Normals defined with *NORMAL override normals defined with *NODE. See "Defining the initial geometry of conventional shell elements," Section 29.6.3 of the Abaqus Analysis User's Manual, for further details.

5.1.4 Reference surface offsets

The reference surface of the shell is defined by the shell element's nodes and normal definitions. When modeling with shell elements, the reference surface is typically coincident with the shell's midsurface. However, many situations arise in which it is more convenient to define the reference surface as offset from the shell's midsurface. For example, surfaces created in CAD packages usually represent either the top or bottom surface of the shell body. In this case it may be easier to define the reference surface to be coincident with the CAD surface and, therefore, offset from the shell's midsurface.

Shell offsets can also be used to define a more precise surface geometry for contact problems where shell thickness is important. Another situation where the offset from the midsurface may be important is when a shell with continuously varying thickness is modeled. In this case defining the nodes at the shell midplane can be difficult. If one surface is smooth while the other is rough, as in some aircraft structures, it is easiest to use shell offsets to define the nodes at the smooth surface.

Offsets can be introduced by specifying an offset value, which is defined as the distance (measured as a fraction of the shell's thickness) from the shell's midsurface to the reference surface containing the element's nodes. Positive values of the offset are in the positive normal direction. When the offset is set equal to 0.5 or SPOS, the top surface of the shell is the reference surface. When the offset is set equal to –0.5 or SNEG, the bottom surface is the reference surface. The default offset is 0, which indicates that the middle surface of the shell is the reference surface. These three reference surface offset settings are shown in Figure 5–4 for a mesh where the nodal positions are adjusted to keep the position of the midsurface constant.

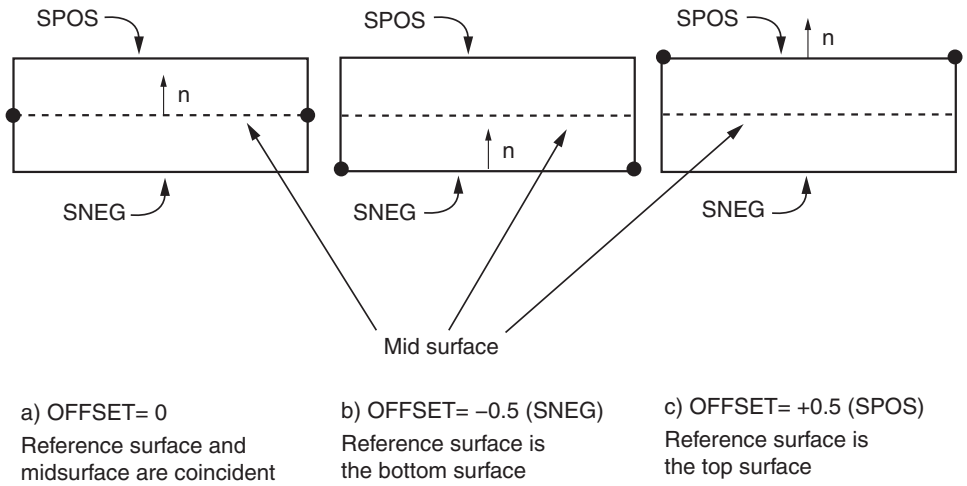


Figure 5–4 Schematic of shell offsets for offset values of 0, –0.5, and +0.5.

The degrees of freedom for the shell are associated with the reference surface. All kinematic quantities, including the element's area, are calculated there. Large offset values for curved shells may lead to a surface integration error, affecting the stiffness, mass, and rotary inertia for the shell section. For stability purposes Abaqus/Explicit also automatically augments the rotary inertia used for shell elements on the order of the offset squared, which may result in errors in the dynamics for large offsets. When large offsets from the shell's midsurface are necessary, use multi-point constraints or rigid body constraints instead.

5.2 Shell formulation – thick or thin

Shell problems generally fall into one of two categories: thin shell problems and thick shell problems. Thick shell problems assume that the effects of transverse shear deformation are important to the solution. Thin shell problems, on the other hand, assume that transverse shear deformation is small enough to be neglected. Figure 5–5(a) illustrates the transverse shear behavior of thin shells: material lines that are initially normal to the shell surface remain straight and normal throughout the deformation. Hence, transverse shear strains are assumed to vanish ($\gamma = 0$). Figure 5–5(b) illustrates the transverse shear behavior of thick shells: material lines that are initially normal to the shell surface do not necessarily remain normal to the surface throughout the deformation, thus adding transverse shear flexibility ($\gamma \neq 0$).

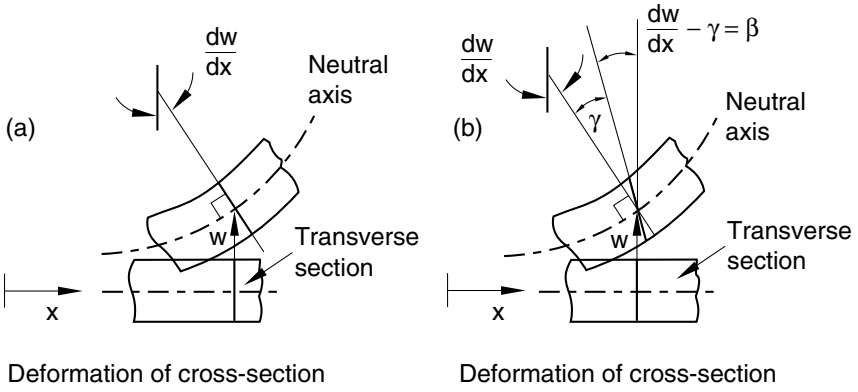


Figure 5–5 Behavior of transverse shell sections in (a) thin shells and (b) thick shells.

Abaqus offers multiple classes of shell elements, distinguished by the element's applicability to thin and thick shell problems. General-purpose shell elements are valid for use with both thick and thin shell problems. In certain cases, for specific applications, enhanced performance can be obtained by using the special-purpose shell elements available in Abaqus/Standard.

The special-purpose shell elements fall into two categories: thin-only shell elements and thick-only shell elements. All special-purpose shell elements provide for arbitrarily large rotations but only small strains. The thin-only shell elements enforce the Kirchhoff constraint; that is, plane sections normal to the midsection of the shell remain normal to the midsurface. The Kirchhoff constraint is enforced either analytically in the element formulation (STRI3) or numerically through the use of a penalty constraint. The thick-only shell elements are second-order quadrilaterals that may produce more accurate results than the general-purpose shell elements in small-strain applications where the loading is such that the solution is smoothly varying over the span of the shell.

To decide if a given application is a thin or thick shell problem, we can offer a few guidelines. For thick shells transverse shear flexibility is important, while for thin shells it is negligible. The significance of transverse shear in a shell can be estimated by its thickness-to-span ratio. A shell made of a single isotropic material with a ratio greater than 1/15 is considered "thick"; if the ratio is less than 1/15, the shell is considered "thin." These estimates are approximate; you should always check the transverse shear effects in your model to verify the assumed shell behavior. Since transverse shear flexibility can be significant in laminated composite shell structures, this ratio should be much smaller for "thin" shell theory to apply. Composite shells with very compliant interior layers (so-called "sandwich" composites) have very low transverse shear stiffness and should almost always be modeled with "thick" shells; if the assumption of plane sections remaining plane is violated, continuum elements should be used. See "Shell section behavior," Section 29.6.4 of the Abaqus Analysis User's Manual, for details on checking the validity of using shell theory.

Transverse shear force and strain are available for general-purpose and thick-only shell elements. For three-dimensional elements, estimates of transverse shear stress are provided. The calculation of

these stresses neglects coupling between bending and twisting deformation and assumes small spatial gradients of material properties and bending moments.

5.3 Shell material directions

Shell elements, unlike continuum elements, use material directions local to each element. Anisotropic material data, such as that for fiber-reinforced composites, and element output variables, such as stress and strain, are defined in terms of these local material directions. In large-displacement analyses the local material axes on a shell surface rotate with the average motion of the material at each integration point.

5.3.1 Default local material directions

The local material 1- and 2-directions lie in the plane of the shell. The default local 1-direction is the projection of the global 1-axis onto the shell surface. If the global 1-axis is normal to the shell surface, the local 1-direction is the projection of the global 3-axis onto the shell surface. The local 2-direction is perpendicular to the local 1-direction in the surface of the shell, so that the local 1-direction, local 2-direction, and the positive normal to the surface form a right-handed set (see Figure 5–6).

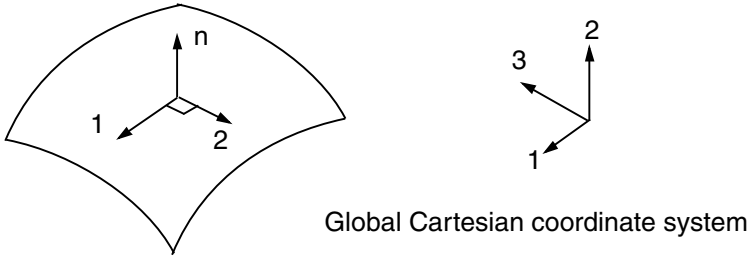


Figure 5–6 Default local shell material directions.

The default set of local material directions can sometimes cause problems; a case in point is the cylinder shown in Figure 5–7.

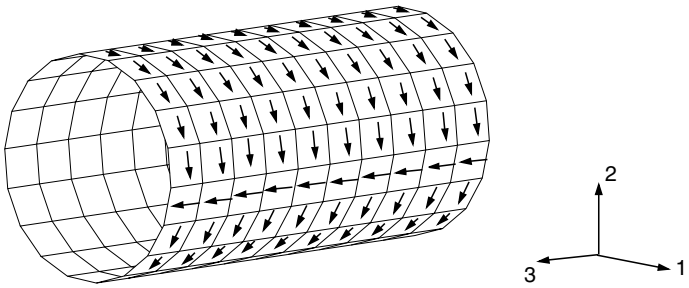


Figure 5–7 Default local material 1-direction in a cylinder.

For most of the elements in the figure the local 1-direction is circumferential. However, there is a line of elements that are normal to the global 1-axis. For these elements the local 1-direction is the projection of the global 3-axis onto the shell, making the local 1-direction axial instead of circumferential. A contour plot of the direct stress in the local 1-direction, σ_{11}, looks very strange, since for most elements σ_{11} is the circumferential stress, whereas for some elements it is the axial stress. In such cases it is necessary to define more appropriate local directions for the model, as discussed in the next section.

5.3.2 Creating alternative material directions

The *ORIENTATION option allows you to control the local material directions directly. With it you can replace the global Cartesian coordinate system with a local rectangular, cylindrical, or spherical coordinate system. You define the orientation of the local (x', y', z') coordinate system by specifying the location of two points, ***a*** and ***b***, as shown in Figure 5–8. For example, a local rectangular system is defined with the following option:

```
*ORIENTATION, SYSTEM=RECTANGULAR, NAME=LOCALR
<x1a>, <x2a>, <x3a>, <x1b>, <x2b>, <x3b>
```

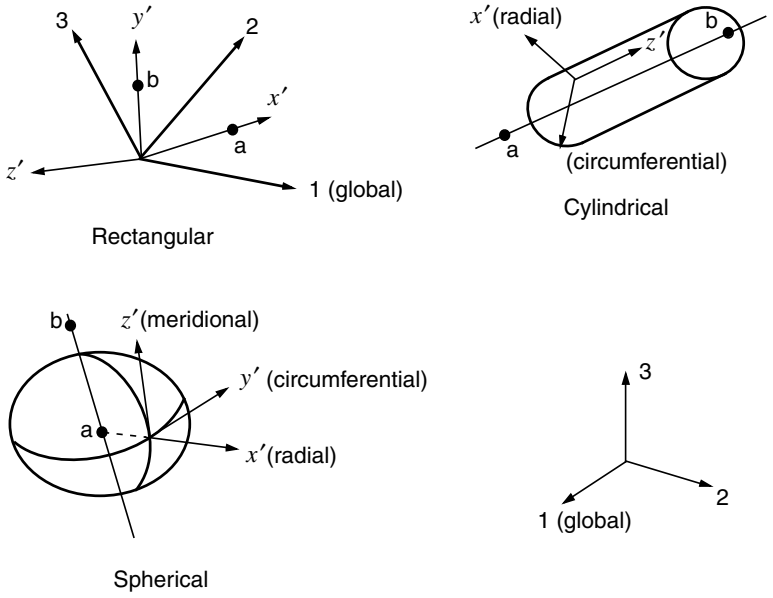


Figure 5–8 Definition of local coordinate systems.

The parameter NAME specifies a label for this orientation, and the coordinates of point *a* (x_1^a, x_2^a, x_3^a) and point b (x_1^b, x_2^b, x_3^b) are given in the global Cartesian system. The local coordinate system is then referred to by the ORIENTATION parameter on the ∗SHELL SECTION or ∗SHELL GENERAL SECTION option.

You must still specify another piece of information. Abaqus must also be told which of the local axes corresponds to which material direction. On the second data line following ∗ORIENTATION, specify the local axis (1, 2, or 3) that is closest to being normal to the shell's surface. Abaqus follows a cyclic permutation (1, 2, 3) of the axes and projects the axis following your selection onto the shell region to form the material 1-direction. For example, if you choose the x'-axis, Abaqus projects the y'-axis onto the shell to form the material 1-direction. The material 2-direction is defined by the cross product of the shell normal and the material 1-direction. Normally, the final material 2-direction and the projection of the other local axis, in this case the z'-axis, will not coincide for curved shells.

If these local axes do not create the desired material directions, you can specify a rotation about the selected axis. The other two local axes are rotated by this amount before they are projected onto the shell's surface to give the final material directions. The following option block would create the local system shown in Figure 5–9:

```
*ORIENTATION, SYSTEM=RECTANGULAR, NAME=LOCALR
<x_1^a>, <x_2^a>, <x_3^a>, <x_1^b>, <x_2^b>, <x_3^b>
1, α
```

Again, it is the rotated y'- and z'-axes that Abaqus projects onto the surface of the shell elements. For the projections to be interpreted easily, the selected axis should be as close as possible to the shell normal.

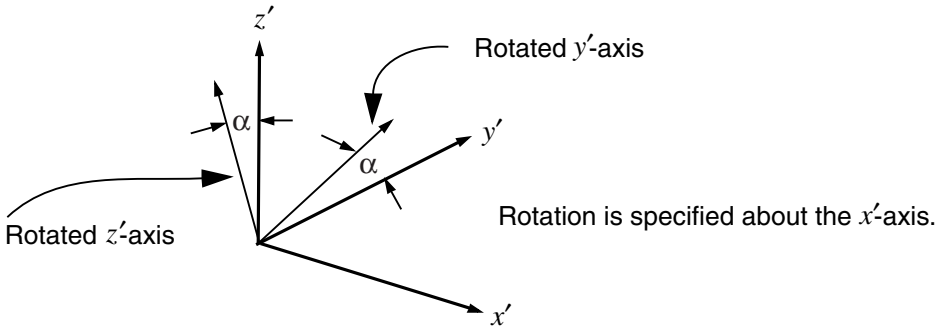


Figure 5–9 Rotation of the local coordinate system for shell elements.

If the centerline of the cylinder shown in Figure 5–7 coincides with the global 3-axis, the following option block could be used to define consistent material directions:

```
*ORIENTATION, SYSTEM=CYLINDRICAL, NAME=CYLIND1
0., 0., 0., 0., 0., 1.
1, 0.
```

Points *a* and *b* lie along the centerline of the cylinder. Since the orientation of the cylinder matches the orientation of our newly defined cylindrical coordinate system, the x'-axis is radial, the y'-axis is circumferential, and the z'-axis is axial. The x'-axis corresponds approximately to the shell normal direction, and a zero rotation is specified; therefore, the projection of the y'-axis onto the shell's surface is the material 1-direction. Thus, the material 1-direction is always circumferential, and the corresponding material 2-direction is always axial.

5.4 Selecting shell elements

- The linear, finite-membrane-strain, fully integrated, quadrilateral shell element (S4) can be used when greater solution accuracy is desired, for problems prone to membrane- or bending-mode hourglassing, or for problems where in-plane bending is expected.

- The linear, finite-membrane-strain, reduced-integration, quadrilateral shell element (S4R) is robust and is suitable for a wide range of applications.
- The linear, finite-membrane-strain, triangular shell elements (S3/S3R) can be used as general-purpose elements. A refined mesh may be needed to capture bending deformations or high strain gradients because of the constant strain approximation in the elements.
- To account for the influence of shear flexibility in laminated composite shell models, use the shell elements suitable for modeling thick shells (S4, S4R, S3/S3R, S8R); check that the assumption of plane sections remaining plane is satisfied.
- Quadratic shell elements, either quadrilateral or triangular, are very effective for general, small-strain, thin-shell applications. These elements are not susceptible to shear or membrane locking.
- If you must use second-order elements in contact simulations, do not use the quadratic, triangular shell element (STRI65). Use the 9-node, quadrilateral shell element (S9R5) instead.
- For very large models that will experience only geometrically linear behavior, the linear, thin-shell element (S4R5) will generally be more cost-effective than the general-purpose shell elements.
- The small membrane strain elements are effective for explicit dynamics problems involving small membrane strains and arbitrarily large rotations.

5.5 Example: skew plate

You have been asked to model the plate shown in Figure 5–10. It is skewed 30° to the global 1-axis, is built-in at one end, and is constrained to move on rails parallel to the plate axis at the other end. You are to determine the midspan deflection when the plate carries a uniform pressure. You are also to assess whether a linear analysis is valid for this problem. You will perform an analysis using Abaqus/Standard.

5.5.1 Coordinate system

The orientation of the structure in the global coordinate system and the suggested origin of the system are shown in Figure 5–10. The plate lies in the global 1–2 plane. Will it be easy to interpret the results of the simulation if you use the default material directions for the shell elements in this model?

5.5.2 Mesh design

Figure 5–11 shows the suggested mesh for this simulation.

You must answer the following questions before selecting an element type: Is the plate thin or thick? Are the strains small or large? The plate is quite thin, with a thickness-to-minimum span ratio of 0.02. (The thickness is 0.8 cm and the minimum span is 40 cm.) While we cannot readily predict the magnitude of the strains in the structure, we think that the strains will be small. Based on this information,

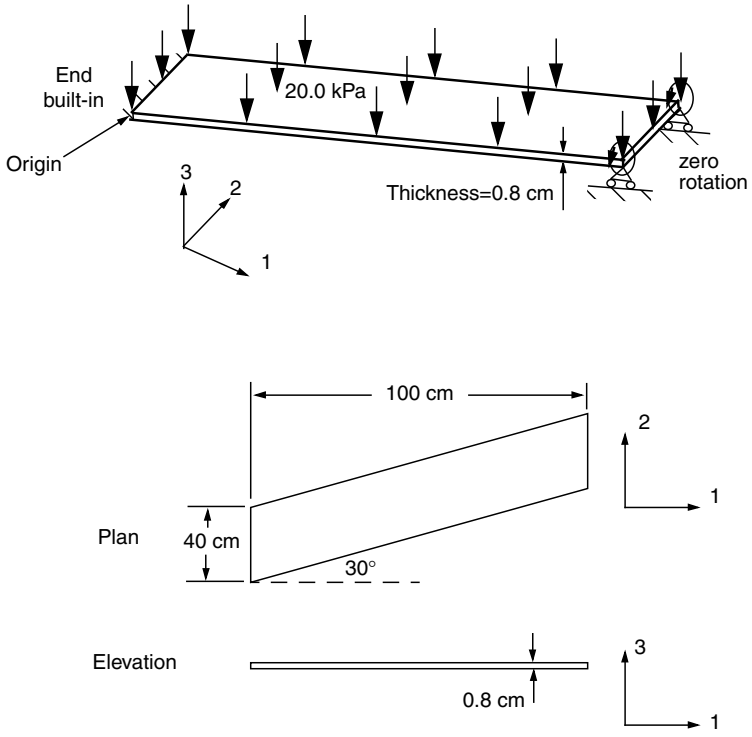


Figure 5–10 Sketch of the skew plate.

you choose quadratic shell elements (S8R5), because they give accurate results for thin shells in small-strain simulations. For further details on shell element selection, consult "Choosing a shell element," Section 29.6.2 of the Abaqus Analysis User's Manual.

5.5.3 Preprocessing—creating the model

The input file for the skew plate example is **`skew.inp`**, which is available in "Skew plate," Section A.3. This example uses the mesh shown in Figure 5–11 by creating the node sets shown in Figure 5–12, and stores all of the the elements in an element set called **`PLATE`**. The following steps in this example describe how the material and history information is defined in this input file. This exercise will give you a better understanding of how the various option blocks combine to define an Abaqus model. If you wish to create the entire model using Abaqus/CAE, refer to "Example: skew plate," Section 5.5 of Getting Started with Abaqus: Interactive Edition.

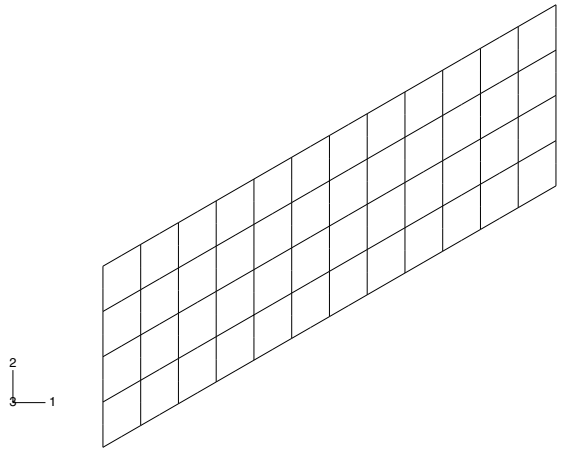


Figure 5–11 Suggested mesh design for the skewed plate simulation.

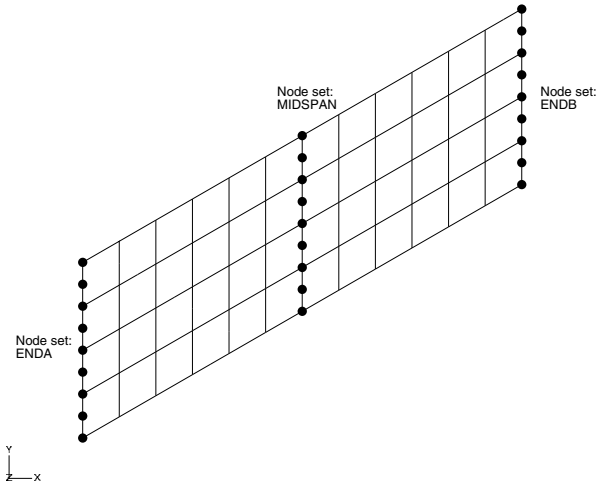


Figure 5–12 Node sets needed for the skew plate simulation.

Before you start to build the model, decide on a system of units. The dimensions are given in cm, but the loading and material properties are given in MPa and GPa. Since these are not consistent units, you must choose a consistent system to use in your model and convert the necessary input data.

5.5.4 Reviewing the input file—the model data

At this point we assume that you have created the basic mesh using your preprocessor. In this section you will review and make corrections to your input file, as well as include additional information, such as material data.

Model description

The following would be a suitable description in the *HEADING option for this simulation:

```
*HEADING
Linear Elastic Skew Plate. 20 kPa Load.
S.I. Units (meters, newtons, sec, kilograms)
```

It clearly explains what you are modeling and what units you are using.

Element connectivity

Check to make sure that you are using the correct element type (S8R5). It is possible that you specified the wrong element type in the preprocessor or that the translator made a mistake when generating the input file. The *ELEMENT option block in your model should begin with the following:

```
*ELEMENT, TYPE=S8R5, ELSET=PLATE
```

In some examples, the name given for the ELSET parameter is not a descriptive name like **PLATE**. If necessary, you may want to change these values, because meaningful names for node and element sets make input files easy to understand.

Node sets

The three node sets shown in Figure 5–12 will be useful in completing the model of the plate. These node sets are described in the input file using *NSET option blocks.

Defining alternative material directions

If you use the default material directions, the direct stress in the material 1-direction, σ_{11}, will contain contributions from both the axial stress, produced by the bending of the plate, and the stress transverse to the axis of the plate. It will be easier to interpret the results if the material directions are aligned with the axis of the plate and the transverse direction. Therefore, a local rectangular coordinate system is needed in which the local x'-direction lies along the axis of the plate (i.e., at 30° to the global 1-axis) and the local y'-direction is also in the plane of the plate.

As you learned in "Shell material directions," Section 5.3, the *ORIENTATION option defines such a local coordinate system. Choose point ***a*** (see Figure 5–8) to have coordinates (10.0E−2, 5.77E−2, 0.)—so that x_2^a/x_1^a = tan 30°—and point *b* to have coordinates (−5.77E−2, 10.E−2, 0.). You must also specify which axis is not projected onto the shell surface (the z'-direction in this

model) as well as an additional rotation (zero using this method). The following *ORIENTATION option block creates the proper local coordinate system, named **SKEW**:

```
*ORIENTATION, NAME=SKEW, SYSTEM=RECTANGULAR
10.0E-2,5.77E-2,0.0, -5.77E-2,10.0E-2,0.0
3, 0.0
```

Alternatively, you can define exactly the same local coordinate system by choosing point *a* and point *b* to lie on the global coordinate 1- and 2-axes and specifying an additional rotation of 30°:

```
*ORIENTATION, NAME=SKEW, SYSTEM=RECTANGULAR
1., 0., 0., 0., 1., 0.
3, 30.
```

Section properties

Since the structure is made of a single material with constant thickness, the section properties are the same for all elements. Therefore, you can use the element set **PLATE** (which includes all elements) to assign the physical and material properties to the elements. Since you assume that the plate is linear elastic, the *SHELL GENERAL SECTION option is more efficient than using the *SHELL SECTION option. The following element property option block defines the section properties for this example:

```
*SHELL GENERAL SECTION, ELSET=PLATE, MATERIAL=MAT1,
ORIENTATION=SKEW
0.8E-2,
```

The ORIENTATION parameter tells Abaqus to use the local coordinate system named **SKEW** to define the material directions for the shells in element set **PLATE**. All element variables will be defined in the **SKEW** coordinate system.

Material properties

The plate is made of an isotropic, linear elastic material that has a Young's modulus of 30.0 GPa and a Poisson's ratio of 0.3. The following material option block specifies this material data:

```
*MATERIAL, NAME=MAT1
*ELASTIC
30.0E9, 0.3
```

Local directions at the nodes

While the *ORIENTATION option defines a local coordinate system for elements, you must use the *TRANSFORM option to define a local coordinate system for nodes. The two options are completely independent of each other. If a node refers to a local coordinate system defined with *TRANSFORM, all data pertaining to the node—such as boundary conditions, concentrated loads, or nodal output variables (displacements, velocities, reaction forces, etc.)—are defined in the transformed coordinate system.

The *TRANSFORM option has the following format:

***TRANSFORM, NSET=**<*node set name*>**, TYPE=**<*axis type*>
<x_1^a**>, <**x_2^a**>, <**x_3^a**>, <**x_1^b**>, <**x_2^b**>, <**x_3^b**>**

The data line specifies the coordinates of two points, ***a*** and ***b***, in much the same way as the *ORIENTATION option. The coordinate system defined with *TRANSFORM does not rotate as the body deforms; it is fixed in the original directions defined at the beginning of the simulation. Rectangular (TYPE=R), cylindrical (TYPE=C), and spherical (TYPE=S) coordinate systems can be specified. Use the NSET parameter to specify the node sets that use this local coordinate system.

As shown in Figure 5–10, one end of the plate is constrained to move on rails that are parallel to the axis of the plate. Since this boundary condition does not coincide with the global axes, you must transform the nodes on this end of the plate into a local coordinate system that has an axis aligned with the plate. The following *TRANSFORM option achieves this transformation:

```
*TRANSFORM, NSET=ENDB, TYPE=R
10.0E-2,5.77E-2,0.0, -5.77E-2,10.0E-2,0.0
```

This option block defines the degrees of freedom for node set **ENDB** in a local coordinate system whose x'-axis is aligned with the long axis of the plate (i.e., the local system is rotated 30° about the global 3-axis).

5.5.5 Reviewing the input file—the history data

We now review the history definition portion of the input file. A single step is needed to define this simulation.

Step definition

The *STEP definition specifies a linear, static simulation:

```
*STEP, PERTURBATION
Uniform pressure (20.0 kPa) load
*STATIC
```

The line following *STEP, PERTURBATION contains a clear description of the loading applied in this step.

Boundary conditions

The nodes at the left-hand end of the plate (node set **ENDA**) need to be constrained completely by the following boundary condition:

```
*BOUNDARY
ENDA, ENCASTRE
```

The nodes at the right-hand end of the plate need to be constrained to model their "rail" boundary condition. Since you have transformed the nodes at this end using *TRANSFORM, you must apply the boundary conditions in the local coordinate system. To allow these nodes to move in the local 1-direction (along the axis of the plate) only, all other degrees of freedom must be constrained as follows:

```
ENDB, 2,6
```

Had you not defined node sets **ENDA** and **ENDB**, you would have had to create a data line for each node.

Loading

A distributed pressure load of 20.0 kPa is applied to the plate in this simulation. As shown in Figure 5–10, the pressure acts in the negative global 3-direction. Pressure loads are applied to the faces of elements with the *DLOAD option (*DLOAD is described in Chapter 4, "Using Continuum Elements," for the lug model example). Shell elements have only one face; therefore, the load identifier for pressure is just "P." A positive pressure on a shell acts in the direction of the positive element normal. The shell elements in the input file from "Skew plate," Section A.3, have normals that align with the positive global 3-axis. Thus, the following input defines the correct pressure loading in that model:

```
*DLOAD
PLATE, P, -20000.0
```

Since element set **PLATE** contains all elements in the model, this option block applies a pressure load to all elements in the model.

Output requests

If the preprocessor has generated default output request options, you should delete them. To create an output database (**.odb**) file for use with Abaqus/Viewer and printed tables of the element stresses, nodal reaction forces and moments, and displacements at the midspan of the plate, the following output requests are included in the input file:

```
*OUTPUT, FIELD, OP=NEW
*NODE OUTPUT
U, RF
*ELEMENT OUTPUT
S, E
*OUTPUT, HISTORY, OP=NEW
*NODE OUTPUT, NSET=MIDSPAN
U,
*EL PRINT
S,
E,
```

```
*NODE PRINT, SUMMARY=NO, TOTALS=YES, GLOBAL=YES
RF,
*NODE PRINT, NSET=MIDSPAN
U,
```

Specifying the *OUTPUT option overrides the default output selections noted in the previous chapters. The option is used with the FIELD and HISTORY parameters to request field and history output to the output database file. In general, field output is used to generate contour plots, symbol plots, and deformed shape plots; history output is used for *X*–*Y* plotting. In conjunction with the *OUTPUT option, the *NODE OUTPUT option is used to request output of nodal variables and the *ELEMENT OUTPUT option is used for output of element variables.

5.5.6 Running the analysis

After storing your input in a file called **skew.inp**, run the analysis interactively. If you do not remember how to run the analysis, see "Running the analysis," Section 4.3.6. If your analysis does not complete, check the data file, **skew.dat**, for error messages. Modify your input file to remove the errors; if you still have trouble running your model, compare your input file to the one given in "Skew plate," Section A.3.

5.5.7 Results

After running the simulation successfully, look at the table of stresses in the data file, **skew.dat**. An excerpt from the table is shown below.

```
   THE FOLLOWING TABLE IS PRINTED FOR ALL ELEMENTS WITH TYPE S8R5 AT THE INTEGRATION POINTS

    ELEMENT   PT SEC FOOT-   S11          S22          S12
                 PT NOTE

           1    1  1  OR -4.2759E+07 -9.3051E+06  6.7584E+06
           1    1  3  OR  4.2759E+07  9.3051E+06 -6.7584E+06
           1    2  1  OR -7.4724E+07 -2.7832E+06  1.0599E+07
           1    2  3  OR  7.4724E+07  2.7832E+06 -1.0599E+07
           1    3  1  OR -7.3273E+07 -2.8832E+07  2.1403E+07
           1    3  3  OR  7.3273E+07  2.8832E+07 -2.1403E+07
           1    4  1  OR -8.2885E+07 -1.8951E+07  1.4786E+07
           1    4  3  OR  8.2885E+07  1.8951E+07 -1.4786E+07
           :
           :
         114    1  1  OR -8.2885E+07 -1.8951E+07  1.4786E+07
         114    1  3  OR  8.2885E+07  1.8951E+07 -1.4786E+07
         114    2  1  OR -7.3273E+07 -2.8832E+07  2.1403E+07
         114    2  3  OR  7.3273E+07  2.8832E+07 -2.1403E+07
         114    3  1  OR -7.4724E+07 -2.7832E+06  1.0599E+07
         114    3  3  OR  7.4724E+07  2.7832E+06 -1.0599E+07
         114    4  1  OR -4.2759E+07 -9.3051E+06  6.7584E+06
         114    4  3  OR  4.2759E+07  9.3051E+06 -6.7584E+06

 MAXIMUM                2.3826E+08  1.0326E+08  7.0025E+07
 ELEMENT                       4           4           4

 MINIMUM               -2.3826E+08 -1.0326E+08 -7.0025E+07
 ELEMENT                       4           4           4

 OR: *ORIENTATION USED FOR THIS ELEMENT
```

The second column (SEC PT—section point) identifies the location in the element where the stress was calculated. Section point 1 lies on the SNEG surface of the shell, and section point 3 lies on the SPOS surface. The letters OR appear in the FOOTNOTE column, indicating that an ∗ORIENTATION option has been used for the element: the stresses refer to a local coordinate system.

Check that the small-strain assumption was valid for this simulation. The axial strain corresponding to the peak stress is $\varepsilon_{11} \approx 0.008$. Because the strain is typically considered small if it is less than 4 or 5%, a strain of 0.8% is well within the appropriate range to be modeled with S8R5 elements.

Look at the reaction forces and moments in the following table:

THE FOLLOWING TABLE IS PRINTED FOR ALL NODES

NODE	FOOT-NOTE	RF1	RF2	RF3	RM1	RM2	RM3
1		0.000	0.000	-109.9	1.775	-0.3283	0.000
2		0.000	0.000	6.448	7.597	-36.46	0.000
3		0.000	0.000	239.9	6.568	-35.46	0.000
4		0.000	0.000	455.4	6.806	-88.26	0.000
5		0.000	0.000	260.5	6.948	-51.13	0.000
6		0.000	0.000	750.8	8.305	-126.5	0.000
7		0.000	0.000	73.90	8.749	-62.23	0.000
8		0.000	0.000	2286.	31.06	-205.8	0.000
9		0.000	0.000	37.19	-1.610	-76.45	0.000
1201		0.000	0.000	37.19	1.610	76.45	0.000
1202		0.000	0.000	2286.	-31.06	205.8	0.000
1203		0.000	0.000	73.90	-8.749	62.23	0.000
1204		0.000	0.000	750.8	-8.305	126.5	0.000
1205		0.000	0.000	260.5	-6.948	51.13	0.000
1206		0.000	0.000	455.4	-6.806	88.26	0.000
1207		0.000	0.000	239.9	-6.568	35.46	0.000
1208		0.000	0.000	6.448	-7.597	36.46	0.000
1209		0.000	0.000	-109.9	-1.775	0.3283	0.000
TOTAL		0.000	0.000	8000.	3.7096E-11	-1.8769E-09	0.000

The reaction forces were written in the global coordinate system because of how we requested the reaction force output (GLOBAL=YES on the ∗NODE PRINT option). Otherwise, the reactions for the nodes would have been written in the local coordinate system. Check that the sum of the reaction forces and reaction moments with the corresponding applied loads is zero. The nonzero reaction force in the 3-direction equilibrates the vertical force of the pressure load (20 kPa × 1.0 m × 0.4 m). In addition to the reaction forces, the pressure load causes self-equilibrating reaction moments at the constrained rotational degrees of freedom.

The table of displacements (which is not shown here) shows that the mid-span deflection across the plate is 5.3 cm, which is approximately 5% of the plate's length. By running this as a linear analysis, we assume the displacements to be small. It is questionable whether these displacements are truly small relative to the dimensions of the structure; nonlinear effects may be important, requiring further investigation. In this case we need to perform a geometrically nonlinear analysis, which is discussed in Chapter 8, "Nonlinearity."

5.5.8 Postprocessing

This section discusses postprocessing with Abaqus/Viewer. Both contour and symbol plots are useful for visualizing shell analysis results. Since contour plotting was discussed in detail in Chapter 4, "Using Continuum Elements," we use symbol plots here.

Start Abaqus/Viewer by typing the following command at the operating system prompt:

```
abaqus viewer odb=skew
```

By default, Abaqus/Viewer plots the undeformed shape of the model.

Element normals

Use the undeformed shape plot to check the model definition. Check that the element normals for the skew-plate model were defined correctly and point in the positive 3-direction.

To display the element normals:

1. From the main menu bar, select **Options→Common**; or use the tool in the toolbox.
   The **Common Plot Options** dialog box appears.
2. Click the **Normals** tab.
3. Toggle on **Show normals**, and accept the default setting of **On elements**.
4. Click **OK** to apply the settings and to close the dialog box.

The default view is isometric. You can change the view using the options in the view menu or the view tools (such as) from the **View Manipulation** toolbar.

To change the view:

1. From the main menu bar, select **View→Specify**.
   The **Specify View** dialog box appears.
2. From the list of available methods, select **Viewpoint**.
3. Enter the X-, Y- and Z-coordinates of the viewpoint vector as `-0.2, -1, 0.8` and the coordinates of the up vector as `0, 0, 1`.
4. Click **OK**.
   Abaqus/Viewer displays your model in the specified view, as shown in Figure 5–13.

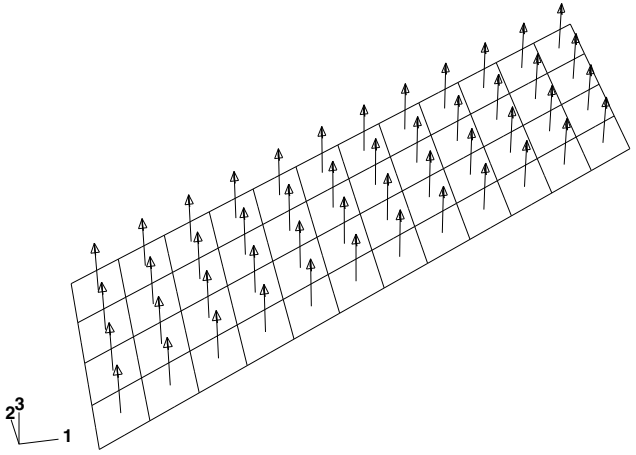


Figure 5–13 Shell element normals in the skew plate model.

Symbol plots

Symbol plots display the specified variable as a vector originating from the node or element integration points. You can produce symbol plots of most tensor- and vector-valued variables. The exceptions are mainly nonmechanical output variables and element results stored at nodes, such as nodal forces. The relative size of the arrows indicates the relative magnitude of the results, and the vectors are oriented along the global direction of the results. The symbol plot legend shows how each arrow color corresponds to a specific range of values. You can plot results for the resultant of variables such as displacement (U), reaction force (RF), etc.; or you can plot individual components of these variables.

Before proceeding, suppress the visibility of the element normals.

To generate a symbol plot of the displacement:

1. From the list of variable types on the left side of the **Field Output** toolbar, select **Symbol**.
2. From the list of output variables in the center of the toolbar, select **U**.
3. From the list of vector quantities and selected components, select **U3**.

   Abaqus/Viewer displays a symbol plot of the displacement vector resultant on the deformed model shape.
4. The default shaded render style obscures the arrows. An unobstructed view of the arrows can be obtained by changing the render style to **Wireframe** using the **Common Plot Options** dialog box. If the element normals are still visible, you should turn them off at this time.
5. The symbol plot can also be based on the undeformed model shape. From the main menu bar, select **Plot→Symbols→On Undeformed Shape**.

A symbol plot on the undeformed model shape appears, as shown in Figure 5–14.

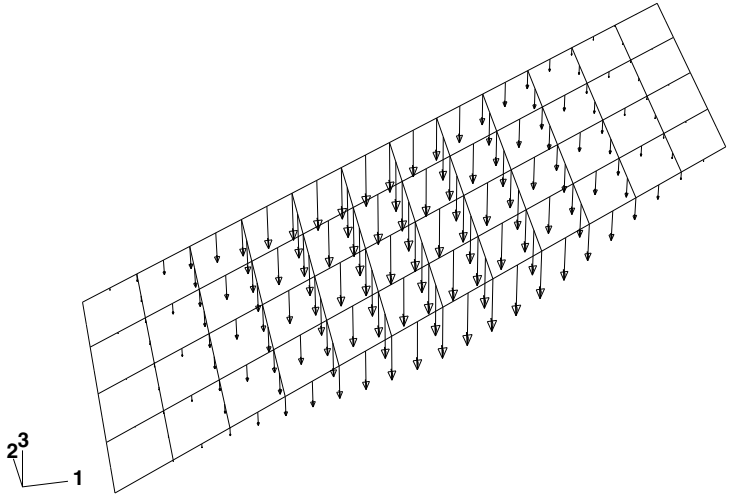


Figure 5–14 Symbol plot of displacement.

You can plot principal values of tensor variables such as stress using symbol plots. A symbol plot of the principal values of stress yields three vectors at every integration point, each corresponding to a principal value oriented along the corresponding principal direction. Compressive values are indicated by arrows pointing toward the integration point, and tensile values are indicated by arrows pointing away from the integration point. You can also plot individual principal values.

To generate a symbol plot of the principal stresses:

1. From the list of variable types on the left side of the **Field Output** toolbar, select **Symbol**.
2. From the list of output variables in the center of the toolbar, select **S**.
3. From the list of tensor quantities and components, select **All principal components** as the tensor quantity.

   Abaqus/Viewer displays a symbol plot of principal stresses.
4. From the main menu bar, select **Options→Symbol**; or use the **Symbol Options** tool in the toolbox to change the arrow length.

   The **Symbol Plot Options** dialog box appears.
5. In the **Color & Style** page, click the **Tensor** tab.
6. Drag the **Size** slider to select `2` as the arrow length.

7. Click **OK** to apply the settings and to close the dialog box.
The symbol plot shown in Figure 5–15 appears.

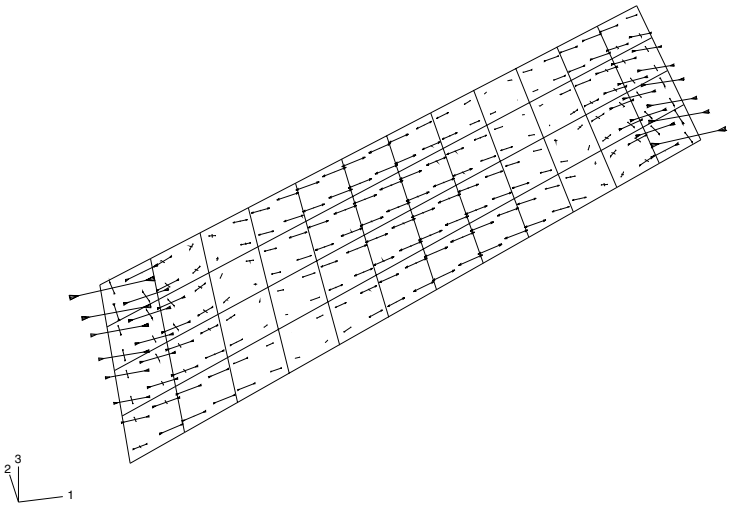


Figure 5–15 Symbol plot of principal stresses on the bottom surface of the plate.

8. The principal stresses are displayed at section point 1 by default. To plot stresses at nondefault section points, select **Result→Section Points** from the main menu bar to open the **Section Points** dialog box.

9. Select the desired nondefault section point for plotting.

10. In a complex model, the element edges can obscure the symbol plots. To suppress the display of the element edges, choose **Feature edges** in the **Basic** tabbed page of the **Common Plot Options** dialog box.

Figure 5–16 shows a symbol plot of the principal stresses at the default section point with only feature edges visible.

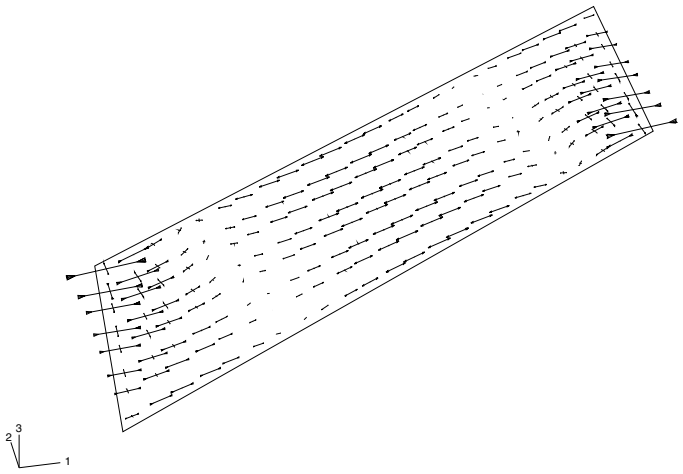


Figure 5–16 Symbol plot of principal stresses using feature edges.

Material directions

Abaqus/Viewer also allows you visualize the element material directions. This feature is particularly useful if you would like to verify that the material directions were assigned correctly in the simulation.

To plot the material directions:

1. From the main menu bar, select **Plot→Material Orientations→On Undeformed Shape**; or use the tool in the toolbox.
   The material orientation directions are plotted on the undeformed shape. By default, the triads that represent the material orientation directions are plotted without arrowheads.
2. From the main menu bar, select **Options→Material Orientation**; or use the **Material Orientation Options** tool in the toolbox to display the triads with arrowheads.
   The **Material Orientation Plot Options** dialog box appears.
3. Set the **Arrowhead** option to use filled arrowheads in the triad.
4. Click **OK** to apply the settings and to close the dialog box.
5. Use the predefined views available in the **Views** toolbar to display the plate as shown in Figure 5–17. In this figure, perspective is turned off. To turn off perspective, click the tool in the **View Options** toolbar.

Tip: If the **Views** toolbar is not visible, select **View→Toolbars→Views** from the main menu bar.

By default, the material 1-direction is colored blue, the material 2-direction is colored yellow, and, if it is present, the material 3-direction is colored red.

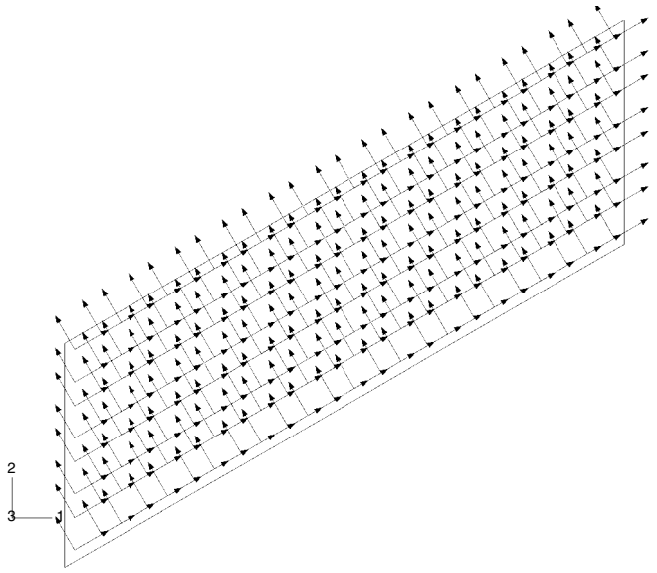


Figure 5–17 Plot of material orientation directions in the plate.

Evaluating results based on tabular data

As noted previously, a convenient alternative to writing printed data to the data (**`.dat`**) file is to generate a tabular report using Abaqus/Viewer. With the aid of display groups, create a tabular data report of the whole model element stresses (components **S11**, **S22**, and **S12**), the reaction forces and moments at the supported nodes (sets **ENDA** and **ENDB**), and the displacements of the midspan nodes (set **MIDSPAN**). The stress data are shown below.

```
Field Output Report

Source 1
---------

   ODB: skew.odb
   Step: Step-1
   Frame: Increment      1: Step Time =   2.2200E-16

Loc 1 : Integration point values at shell general ... : SNEG, (fraction = -1.0)
Loc 2 : Integration point values at shell general ... : SPOS, (fraction =  1.0)

Output sorted by column "Element Label".

Field Output reported at integration points for part: PLATE-1
```

Element Label	Int Pt	S.S11 @Loc 1	S.S11 @Loc 2	S.S22 @Loc 1	S.S22 @Loc 2	S.S12 @Loc 1	S.S12 @Loc 2
1	1	-42.7593E+06	42.7593E+06	-9.30515E+06	9.30515E+06	6.75836E+06	-6.75836E+06
1	2	-74.7242E+06	74.7242E+06	-2.78322E+06	2.78322E+06	10.5987E+06	-10.5987E+06
1	3	-73.2731E+06	73.2731E+06	-28.832E+06	28.832E+06	21.4032E+06	-21.4032E+06
1	4	-82.8849E+06	82.8849E+06	-18.9513E+06	18.9513E+06	14.7861E+06	-14.7861E+06
.							
.							
114	1	-82.8849E+06	82.8849E+06	-18.9513E+06	18.9513E+06	14.7861E+06	-14.7861E+06
114	2	-73.2731E+06	73.2731E+06	-28.832E+06	28.832E+06	21.4032E+06	-21.4032E+06
114	3	-74.7242E+06	74.7242E+06	-2.78322E+06	2.78322E+06	10.5987E+06	-10.5987E+06
114	4	-42.7593E+06	42.7593E+06	-9.30515E+06	9.30515E+06	6.75836E+06	-6.75836E+06
Minimum		-238.256E+06	-90.2214E+06	-103.26E+06	-10.5215E+06	-18.8595E+06	-70.0247E+06
At Element		4	54	4	63	81	111
Int Pt		3	3	1	1	2	2
Maximum		90.2214E+06	238.256E+06	10.5215E+06	103.26E+06	70.0247E+06	18.8595E+06
At Element		54	4	63	4	111	81
Int Pt		3	3	1	1	2	2

The reaction forces and moments are listed in the following table:

```
Field Output Report

Source 1
---------

   ODB: skew.odb
   Step: Step-1
   Frame: Increment      1: Step Time =   2.2200E-16

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: PART-1-1
```

Node Label	RF.RF1 @Loc 1	RF.RF2 @Loc 1	RF.RF3 @Loc 1	RM.RM1 @Loc 1	RM.RM2 @Loc 1	RM.RM3 @Loc 1
1	0.	0.	-109.912	1.77484	-328.266E-03	0.
2	0.	0.	6.44824	7.59742	-36.4615	0.
3	0.	0.	239.923	6.5683	-35.4597	0.
4	0.	0.	455.379	6.80581	-88.2614	0.
5	0.	0.	260.543	6.94783	-51.1276	0.
6	0.	0.	750.833	8.30465	-126.458	0.
7	0.	0.	73.904	8.74902	-62.2273	0.
8	0.	0.	2.28569E+03	31.0634	-205.759	0.
9	0.	0.	37.1932	-1.6098	-76.4492	0.
1201	0.	0.	37.1932	1.6098	76.4492	0.
1202	0.	0.	2.28569E+03	-31.0634	205.759	0.
1203	0.	0.	73.904	-8.74902	62.2273	0.
1204	0.	0.	750.833	-8.30465	126.458	0.
1205	0.	0.	260.543	-6.94783	51.1276	0.
1206	0.	0.	455.379	-6.80581	88.2614	0.
1207	0.	0.	239.923	-6.5683	35.4597	0.
1208	0.	0.	6.44824	-7.59742	36.4615	0.
1209	0.	0.	-109.912	-1.77484	328.266E-03	0.
Minimum	0.	0.	-109.912	-31.0634	-205.759	0.
At Node	1209	1209	1	1202	8	1209
Maximum	0.	0.	2.28569E+03	31.0634	205.759	0.
At Node	1209	1209	8	8	1202	1209
Total	0.	0.	8.00000E+03	0.	0.	0.

5.6 Related Abaqus examples

- "Pressurized fuel tank with variable shell thickness," Section 2.1.6 of the Abaqus Example Problems Manual
- "Analysis of an anisotropic layered plate," Section 1.1.2 of the Abaqus Benchmarks Manual
- "Buckling of a simply supported square plate," Section 1.2.4 of the Abaqus Benchmarks Manual
- "The barrel vault roof problem," Section 2.3.1 of the Abaqus Benchmarks Manual

5.7 Suggested reading

The following references provide a more in-depth treatment of the theoretical and computational aspects of shell theory.

Basic shell theory

- Timoshenko, S., *Strength of Materials: Part II*, Krieger Publishing Co., 1958.
- Timoshenko, S. and S. W. Krieger, *Theory of Plates and Shells*, McGraw-Hill, Inc., 1959.
- Ugural, A. C., *Stresses in Plates and Shells*, McGraw-Hill, Inc., 1981.

Basic computational shell theory

- Cook, R. D., D. S. Malkus, and M. E. Plesha, *Concepts and Applications of Finite Element Analysis*, John Wiley & Sons, 1989.
- Hughes, T. J. R., *The Finite Element Method*, Prentice-Hall, Inc., 1987.

Advanced shell theory

- Budiansky, B., and J. L. Sanders, "On the 'Best' First-Order Linear Shell Theory," Progress in Applied Mechanics, The Prager Anniversary Volume, 129–140, 1963.

Advanced computational shell theory

- Ashwell, D. G., and R. H. Gallagher, *Finite Elements for Thin Shells and Curved Members*, John Wiley & Sons, 1976.
- Hughes, T. J. R., T. E. Tezduyar, "Finite Elements Based upon Mindlin Plate Theory with Particular Reference to the Four-Node Bilinear Isoparametric Element," Journal of Applied Mechanics, 587–596, 1981.
- Simo, J. C., D. D. Fox, and M. S. Rifai, "On a Stress Resultant Geometrically Exact Shell Model. Part III: Computational Aspects of the Nonlinear Theory," Computer Methods in Applied Mechanics and Engineering, vol. 79, 21–70, 1990.

5.8 Summary

- The cross-section behavior of shell elements can be determined using numerical integration through the shell thickness (*SHELL SECTION) or using a cross-section stiffness calculated at the beginning of the analysis (*SHELL GENERAL SECTION).
- *SHELL GENERAL SECTION is efficient, but it can be used only with linear materials. *SHELL SECTION can be used with both linear and nonlinear materials.
- Numerical integration is performed at a number of section points through the shell thickness. These section points are the locations at which element variables can be output. The default outermost section points lie on the surfaces of the shell.
- The direction of a shell element's normal determines the positive and negative surfaces of the element. To define contact and interpret element output correctly, you must know which surface is which. The shell normal also defines the direction of positive pressure loads applied to the element and can be plotted in Abaqus/Viewer.
- Shell elements use material directions local to each element. In large-displacement analyses the local material axes rotate with the element. *ORIENTATION can be used to define non-default local coordinate systems. The element variables, such as stress and strain, are output in the local directions.
- *TRANSFORM defines local coordinate systems for nodes. Concentrated loads and boundary conditions are applied in the local coordinate system. All printed nodal output, such as displacements, also refer to the local system by default.
- Symbol plots can help you visualize the results from a simulation. They are especially useful for visualizing the motion and load paths of a structure.

6. Using Beam Elements

Use beam elements to model structures in which one dimension (the length) is significantly greater than the other two dimensions and in which the longitudinal stress is most important. Beam theory is based on the assumption that the deformation of the structure can be determined entirely from variables that are functions of position along the structure's length. For beam theory to produce acceptable results, the cross-section dimensions should be less than 1/10 of the structure's typical axial dimension. The following are examples of typical axial dimensions:

- the distance between supports,
- the distance between gross changes in cross-section, and
- the wavelength of the highest vibration mode of interest.

Abaqus beam elements assume that plane sections perpendicular to the axis of the beam remain plane during deformation.

Do not be confused into thinking that the cross-section dimensions should be less than 1/10 of a typical *element* length. A highly refined mesh may contain beam elements whose length is less than their cross-section dimensions, although this is not generally recommended—continuum elements may be more suitable in such a case.

6.1 Beam cross-section geometry

You can use the *BEAM SECTION option or the *BEAM GENERAL SECTION option to define the beam section. With either option you can define the beam cross-section geometrically by specifying the shape and dimensions of the section. The *BEAM GENERAL SECTION option can also be used to define the beam section through section engineering properties, such as area and moments of inertia. Alternatively, the beam section can be based on a mesh of special two-dimensional elements for which geometric quantities are calculated numerically.

Abaqus offers a variety of common cross-section shapes, as shown in Figure 6–1, should you decide to define the beam profile geometrically. You can also define almost any thin-walled cross-section using the arbitrary cross-section definition. For a detailed discussion of the beam cross-sections available in Abaqus, see "Beam cross-section library," Section 29.3.9 of the Abaqus Analysis User's Manual.

The basic format of the *BEAM SECTION option is

***BEAM SECTION, ELSET=**<*element set name*>**, SECTION=**<*section type*>**,**
MATERIAL=<*material name*>
<*cross-section dimensions*>
$<n_1^1>$,$<n_1^2>$,$<n_1^3>$

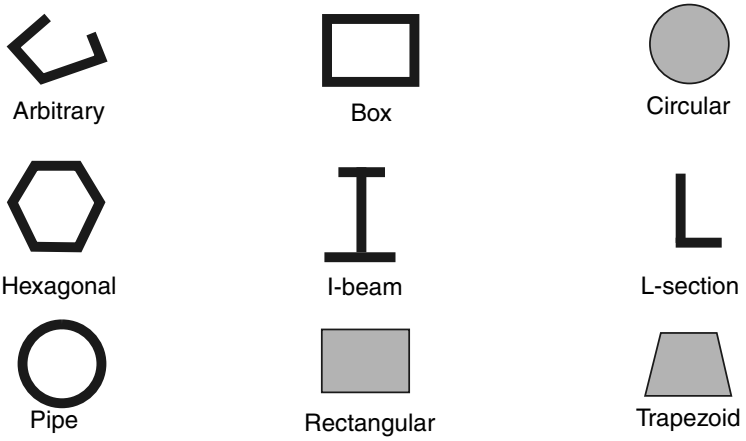


Figure 6–1 Beam cross-sections.

Set the SECTION parameter to one of the cross-sections shown in Figure 6–1. Provide the required cross-section dimensions, which are different for each type of cross-section, as specified in "Beam cross-section library," Section 29.3.9 of the Abaqus Analysis User's Manual. The vector on the second data line defines the approximate normal, $\mathbf{n}_1$, which is explained later in this section.

The basic format of the *BEAM GENERAL SECTION option is

***BEAM GENERAL SECTION, ELSET=**<*element set name*>**, SECTION=**<*section type*>
<*cross-section dimensions*> *or* <*section engineering properties*>
$<n_1^1>$**,**$<n_1^2>$**,**$<n_1^3>$
<*Young's modulus (E)*>**,**<*torsional shear modulus (G)*>

To define the section's properties geometrically, set the SECTION parameter to one of the cross-sections shown in Figure 6–1. In this case you provide the required cross-section dimensions the same way as you would with *BEAM SECTION. The vector on the second data line again defines the approximate normal, $\mathbf{n}_1$. On the third line you enter the elastic material constants, because *BEAM GENERAL SECTION does not refer to any material option block. If you define the section's properties geometrically with this option, the material behavior must be linear elastic.

The alternative is to set the SECTION parameter to GENERAL or NONLINEAR GENERAL, in which case you provide the section engineering properties (area, moments of inertia, and torsional constants) instead of the cross-section dimensions. These parameters allow you to combine the beam's geometry and material behavior to define its response to loads. This response may be linear or nonlinear. See "Using a general beam section to define the section behavior," Section 29.3.7 of the Abaqus Analysis User's Manual, for further details.

In Abaqus/Standard you can also define beams with linearly tapered cross-sections. General beam sections with linear response and standard library sections are supported.

Meshed beam cross-sections allow a description of the beam cross-section that includes multiple materials and complex geometry. This type of beam profile is discussed further in "Meshed beam cross-sections," Section 10.6.1 of the Abaqus Analysis User's Manual.

6.1.1 Section points

When you specify *BEAM SECTION, Abaqus calculates the beam element's response at an array of section points throughout the beam cross-section. The number of section points, as well as the section point locations, are shown in "Beam cross-section library," Section 29.3.9 of the Abaqus Analysis User's Manual. Element output variables, such as stress and strain, are available at any of the section points; however, by default output is provided at only a select number of section points, as listed in "Beam cross-section library," Section 29.3.9 of the Abaqus Analysis User's Manual. All the section points for a rectangular cross-section (SECTION=RECT) are shown in Figure 6–2.

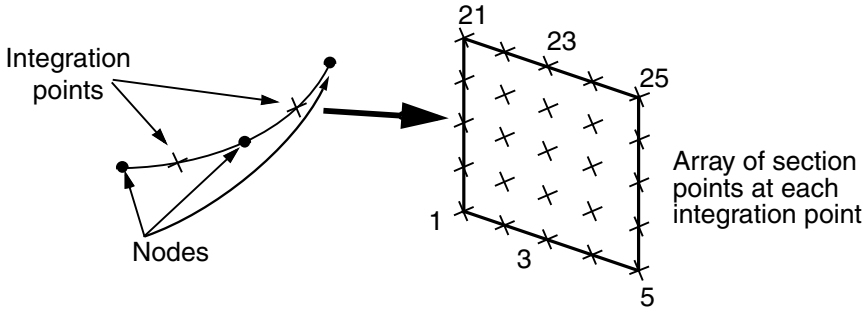


Figure 6–2 Integration and default section points in a B32 rectangular beam element.

For this cross-section output is provided at points 1, 5, 21, and 25 by default. The beam element shown in Figure 6–2 uses a total of 50 section points, 25 at each of the two integration points, to calculate its stiffness.

When you specify *BEAM GENERAL SECTION, Abaqus does not calculate the beam's response at the section points. Instead, it uses the section engineering properties to determine the section response. Therefore, Abaqus uses section points only as locations for output, and you need to specify the section points at which you desire output. Use the *SECTION POINTS option, which must follow the *BEAM GENERAL SECTION option, to specify the location of the section points:

```
*SECTION POINTS
<x1>,<x2>
```

The x_1- and x_2-coordinates are given in the local 1–2 coordinate system of the beam cross-section. For example, if we require stresses at the corners of an element with the rectangular beam cross-section shown in Figure 6–3, we would use the following option block:

```
*SECTION POINTS
-0.01, -0.005
 0.01, -0.005
 0.01,  0.005
-0.01,  0.005
```

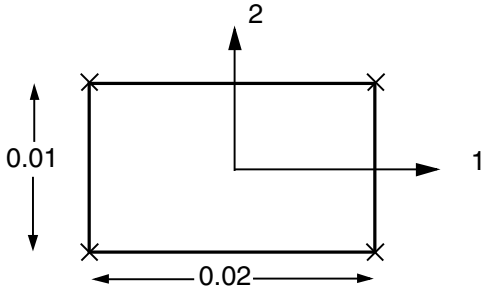


Figure 6–3 Section points at the corners of a rectangular beam.

The points you specify are assigned identifying numbers based on the order they are given; i.e., the first point is section point 1, the second is section point 2, etc.

6.1.2 Cross-section orientation

You must define the orientation of a beam's cross-section in global Cartesian space. The local tangent along the beam element, $\mathbf{t}$, is defined as the vector along the element axis pointing from the first node of the element to the next node. The beam cross-section is perpendicular to this local tangent. The local (1–2) beam section axes are represented by the vectors $\mathbf{n}_1$ and $\mathbf{n}_2$. The three vectors $\mathbf{t}$, $\mathbf{n}_1$, and $\mathbf{n}_2$ form a local, right-handed, coordinate system (see Figure 6–4).

The $\mathbf{n}_1$-direction is always (0.0, 0.0, −1.0) for two-dimensional beam elements.

For three-dimensional beam elements there are several ways to define the orientation of the local beam section axes. The simplest is to specify an extra node on the data line defining the element in the *ELEMENT option. The vector, $\mathbf{v}$, from the first node in the beam element to this additional node (see Figure 6–4), is used initially as an approximate $\mathbf{n}_1$-direction. Abaqus then defines the beam's $\mathbf{n}_2$-direction as $\mathbf{t} \times \mathbf{v}$. Having determined $\mathbf{n}_2$, Abaqus defines the actual $\mathbf{n}_1$-direction as $\mathbf{n}_2 \times \mathbf{t}$. This procedure ensures that the local tangent and local beam section axes form an orthogonal system.

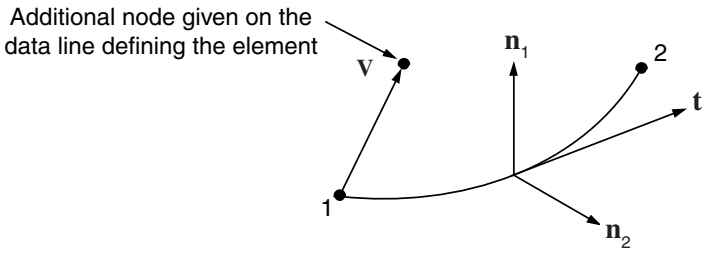


Figure 6–4 Orientation of the beam element tangent, $\mathbf{t}$, and beam section axes, $\mathbf{n}_1$ and $\mathbf{n}_2$.

Alternatively, you can give an approximate $\mathbf{n}_1$-direction on the element section option (either *BEAM SECTION or *BEAM GENERAL SECTION). Abaqus then uses the procedure described above to calculate the actual beam section axes. If you specify both an extra node and an approximate $\mathbf{n}_1$-direction, the additional node method takes precedence. Abaqus uses the vector from the origin to the point (0.0, 0.0, −1.0) as the default $\mathbf{n}_1$-direction if you provide no approximate $\mathbf{n}_1$-direction.

There are two methods that can be used to override the $\mathbf{n}_2$-direction defined by Abaqus. One is to give the components of $\mathbf{n}_2$ as the 4th, 5th, and 6th data values following the nodal coordinates on the data lines of the *NODE option. The alternative is to use the *NORMAL option. If both methods are used, the *NORMAL option takes precedence. Abaqus again defines the $\mathbf{n}_1$-direction as $\mathbf{n}_2 \times \mathbf{t}$.

The $\mathbf{n}_2$-direction that you provide need not be orthogonal to the beam element tangent, $\mathbf{t}$. When you provide the $\mathbf{n}_2$-direction, the local beam element tangent is redefined as the value of the cross product $\mathbf{n}_1 \times \mathbf{n}_2$. It is quite possible in this situation that the redefined local beam tangent, $\mathbf{t}$, will not align with the beam axis, as defined by the vector from the first to the second node. If the $\mathbf{n}_2$-direction subtends an angle greater than 20° with the plane perpendicular to the element axis, Abaqus issues a warning message in the data file.

The example presented in "Example: cargo crane," Section 6.4, explains how to assign the beam cross-section orientation in your model.

6.1.3 Beam element curvature

The curvature of beam elements is based on the orientation of the beam's $\mathbf{n}_2$-direction relative to the beam axis. If the $\mathbf{n}_2$-direction and the beam axis are not orthogonal (i.e., the beam axis and the tangent, $\mathbf{t}$, do not coincide), the beam element is considered to be curved initially. Since the behavior of curved beams is different from the behavior of straight beams, you should always check your model to ensure that the correct normals and, hence, the correct curvatures are used. For beams and shells Abaqus uses the same algorithm to determine the normals at nodes shared by several elements. A description is given in "Beam element cross-section orientation," Section 29.3.4 of the Abaqus Analysis User's Manual.

If you intend to model curved beam structures, you should use one of the two methods described earlier to define the $\mathbf{n}_2$-direction directly, allowing you great control in modeling the curvature. Even

if you intend to model a structure made up of straight beams, curvature may be introduced as normals are averaged at shared nodes. You can rectify this problem by defining the beam normals directly as explained previously.

6.1.4 Nodal offsets in beam sections

When beam elements are used as stiffeners for shell models, it is convenient to have the beam and shell elements share the same nodes. By default, shell element nodes are located at the midplane of the shell, and beam element nodes are located somewhere in the cross-section of the beam. Hence, if the shell and beam elements share the same nodes, the shell and the beam stiffener will overlap unless the beam cross-section is offset from the location of the node (see Figure 6–5).

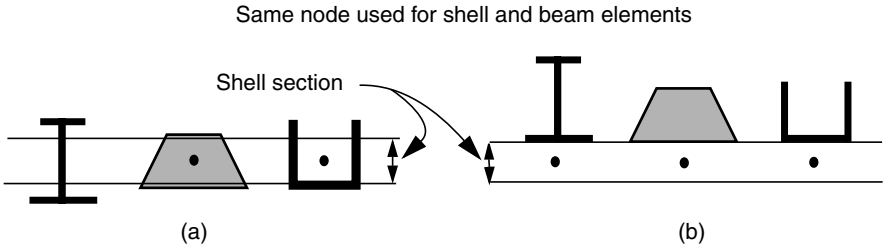


Figure 6–5 Using beams as stiffeners for shell models: (a) without offset of beam sections; (b) with offset of beam sections.

With beam section types I, TRAPEZOID, and ARBITRARY it is possible to specify that the section geometry is located at some distance from the origin of the section's local coordinate system, which is located at the element's nodes. Since it is easy to offset beams with such cross-sections from their nodes, they can be used readily as stiffeners as shown in Figure 6–5(b). (If flange or web buckling of the stiffeners is important, shells should be used to model the stiffeners.)

The I-beam shown in Figure 6–6 is attached to a shell 1.2 units thick. The following input is used to orient the beam section as it is shown in the figure:

```
*BEAM SECTION, SECTION=I, ELSET=<element set name>, MATERIAL=<material>
-0.6, 2.4, 3.0, 2.0, 0.2, 0.2, 0.2
<n1^1>, <n1^2>,<n1^3>
```

The first item on the first data line defines the offset of the beam node from the bottom of the I-section. The offset is one half of the shell thickness or 0.6. The remaining data items are the beam depth, the width of the bottom and top flanges, the thickness of the bottom and top flanges, and the thickness of the web.

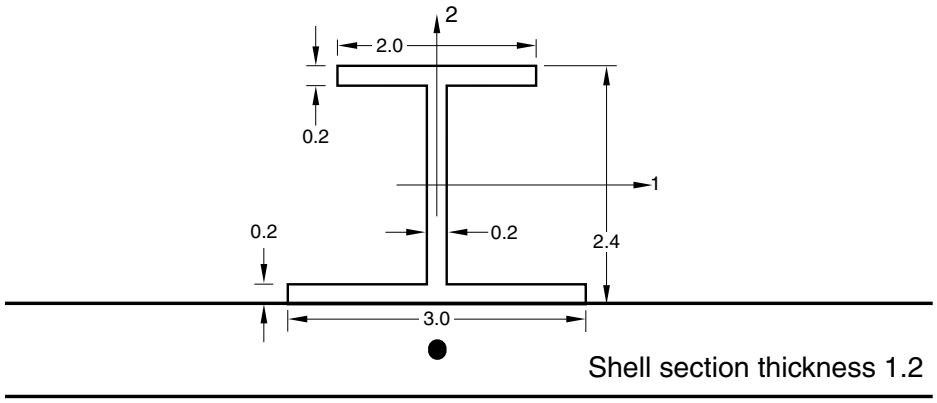


Figure 6–6 I-beam used as stiffener for a shell element.

You can give the location of the centroid and shear center if you specify the ∗BEAM GENERAL SECTION option with the parameter SECTION=GENERAL. The ∗SHEAR CENTER and ∗CENTROID options allow these locations to be offset from the node, enabling you to model stiffeners readily. For example, the input for the I-beam attached to the shell as shown in Figure 6–6 is:

```
*BEAM GENERAL SECTION, SECTION=GENERAL, ELSET=<element set name>
1.4, 1.312, 0., 0.585, 0.0192 ◄— Area, I11, I12, I22, torsional rigidity
<n1 1>, <n1 2>, <n1 3>
<Young's modulus>,<torsional shear modulus>
*CENTROID
0., 1.642857
*SHEAR CENTER
0., 1.202857
```

It is also possible to define the beam nodes and shell nodes separately and connect the beam and shell using a rigid beam constraint between the two nodes. See “Linear constraint equations,” Section 34.2.1 of the Abaqus Analysis User’s Manual, for further details.

6.2 Formulation and integration

All beam elements in Abaqus are “beam-column” elements—meaning they allow axial, bending, and torsional deformation. The Timoshenko beam elements also consider the effects of transverse shear deformation.

6.2.1 Shear deformation

The linear elements (B21 and B31) and the quadratic elements (B22 and B32) are shear deformable, Timoshenko beams; thus, they are suitable for modeling both stout members, in which shear deformation is important, and slender beams, in which shear deformation is not important. The cross-sections of these elements behave in the same manner as the cross-sections of the thick shell elements, as illustrated in Figure 6–7(b) and discussed in "Shell formulation – thick or thin," Section 5.2.

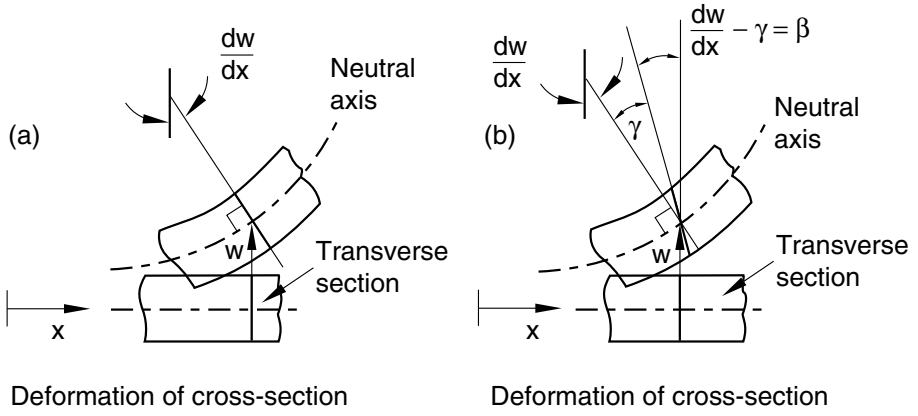


Figure 6–7 Behavior of transverse beam sections in (a) slender beams and (b) thick beams.

Abaqus assumes the transverse shear stiffness of these beam elements to be linear elastic and constant. In addition, these beams are formulated so that their cross-sectional area can change as a function of the axial deformation, an effect that is considered only in geometrically nonlinear simulations (see Chapter 8, "Nonlinearity") in which the POISSON parameter on the beam section property option has a nonzero value. These elements can provide useful results as long as the cross-section dimensions are less than 1/10 of the typical axial dimensions of the structure, which is generally considered to be the limit of the applicability of beam theory; if the beam cross-section does not remain plane under bending deformation, beam theory is not adequate to model the deformation.

The cubic elements available in Abaqus/Standard—the so-called Euler-Bernoulli beam elements (B23 and B33)—do not model shear flexibility. The cross-sections of these elements remain perpendicular to the beam axis (see Figure 6–7(a)). Therefore, the cubic beam elements are most effective for modeling structures with relatively slender members. Since cubic elements model a cubic variation of displacement along their lengths, a structural member often can be modeled with a single cubic element for a static analysis and with a small number of elements for a dynamic analysis. These elements assume that shear deformations are negligible. Generally, if the cross-section dimensions are less than 1/15 of the typical axial dimensions of the structure, this assumption is valid.

6.2.2 Torsional response—warping

Structural members are often subjected to torsional moments, which occur in almost any three-dimensional frame structure. Loads that cause bending in one member may cause twisting in another, as shown in Figure 6–8.

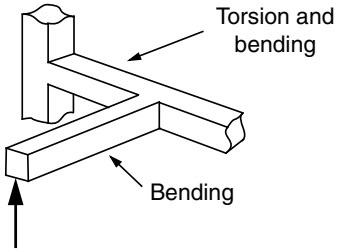


Figure 6–8 Torsion induced in a frame structure.

The response of a beam to torsion depends on the shape of its cross-section. Generally, torsion in a beam produces warping or nonuniform out-of-plane displacements in the cross-section. Abaqus considers the effects of torsion and warping only in the three-dimensional elements. The warping calculation assumes that the warping displacements are small. The following cross-sections behave differently under torsion: solid cross-sections; closed, thin-walled cross-sections; and open, thin-walled cross-sections.

Solid cross-sections

A solid, non-circular cross-section does not remain plane under torsion; instead, the section warps. Abaqus uses St. Venant warping theory to calculate a single component of shear strain caused by the warping at each section point in the cross-section. The warping in such solid cross-sections is considered unconstrained and creates negligible axial stresses. (Warping constraints would affect the solution only in the immediate vicinity of the constrained end.) The torsional stiffness of a beam with a solid cross-section depends on the shear modulus, G, of the material and the torsion constant, J, of the beam section. The torsion constant depends on the shape and the warping characteristics of the beam cross-section. Torsional loads that produce large amounts of inelastic deformation in the cross-section cannot be modeled accurately with this approach.

Closed, thin-walled cross-sections

Beams that have closed, thin-walled, non-circular cross-sections (BOX or HEX) have significant torsional stiffness and, thus, behave in a manner similar to solid sections. Abaqus assumes that warping in these sections is also unconstrained. The thin-walled nature of the cross-section allows Abaqus to consider the shear strains to be constant through the wall thickness. The thin-walled assumption is generally valid provided that the wall thickness is 1/10 a typical beam cross-section dimension. Examples of typical cross-section dimensions for thin-walled cross-sections include:

- The diameter of a pipe section.
- The length of an edge of a box section.
- The typical edge length of an arbitrary section.

Open, thin-walled cross-sections

Open, thin-walled cross-sections are very flexible in torsion when warping is unconstrained, and the primary source of torsional stiffness in such structures is the constraint of the axial warping strains. Constraining the warping of open, thin-walled beams introduces axial stresses that can affect the beam's response to other loading types. Abaqus/Standard has shear deformable beam elements, B31OS and B32OS, which include the warping effects in open, thin-walled sections. These elements must be used when modeling structures with open, thin-walled cross-sections—such as a channel (defined as an ARBITRARY section) or an I-section—that are subjected to significant torsional loading.

The variation of the warping-induced axial deformation over the beam's cross-section is defined by the section's warping function. The magnitude of this function is treated as an extra degree of freedom, 7, in the open-section beam elements. Constraining this degree of freedom prevents warping at the nodes at which the constraints are applied.

So that the warping amplitude can be different in each branch, the junction between open-section beams in a frame structure generally should be modeled with separate nodes for each branch (see Figure 6–9).

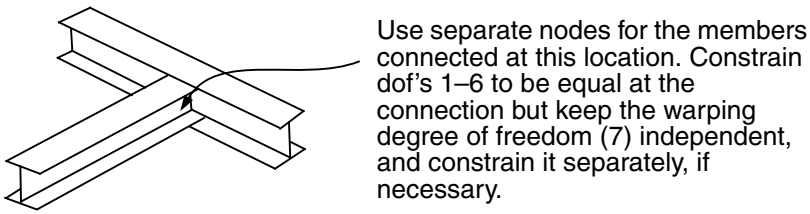

Use separate nodes for the members connected at this location. Constrain dof's 1–6 to be equal at the connection but keep the warping degree of freedom (7) independent, and constrain it separately, if necessary.

Figure 6–9 Connecting open-section beams.

However, if the connection is designed to prevent warping, all branches should share a common node, and the warping degree of freedom should be constrained using the *BOUNDARY option.

A shear force that does not act through the beam's shear center produces torsion. The twisting moment is equal to the shear force multiplied by its eccentricity with respect to the shear center. Often, the centroid and the shear center do not coincide in open, thin-walled beam sections (see Figure 6–10). If the nodes are not located at the shear center of the cross-section, the section may twist under loading.

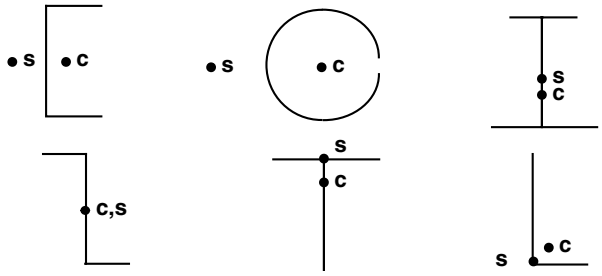


Figure 6–10 Approximate locations of shear centers, *s*, and centroids, *c*, for a number of beam cross-sections.

6.3 Selecting beam elements

- First-order, shear-deformable beam elements (B21, B31) should be used in any simulation that includes contact.
- If the transverse shear deformation is important, use Timoshenko (quadratic) beam elements (B22, B32).
- If the structure is either very rigid or very flexible, the hybrid beam elements (B21H, B32H, etc.) available in Abaqus/Standard should be used in geometrically nonlinear simulations.
- The Euler-Bernoulli (cubic) beams (B23, B33) available in Abaqus/Standard are very accurate for simulations that include distributed loading, such as dynamic vibration analyses.
- Structures with open, thin-walled cross-sections should be modeled with the elements that use open-section warping theory (B31OS, B32OS) available in Abaqus/Standard.

6.4 Example: cargo crane

A light-service, cargo crane is shown in Figure 6–11. You have been asked to determine the static deflections of the crane when it carries a load of 10 kN. You should also identify the critical members and joints in the structure: i.e., those with the highest stresses and loads. Because this is a static analysis you will analyze the cargo crane using Abaqus/Standard.

The crane consists of two truss structures joined together by cross bracing. The two main members in each truss structure are steel box beams (box cross-sections). Each truss structure is stiffened by internal bracing, which is welded to the main members. The cross bracing connecting the two truss

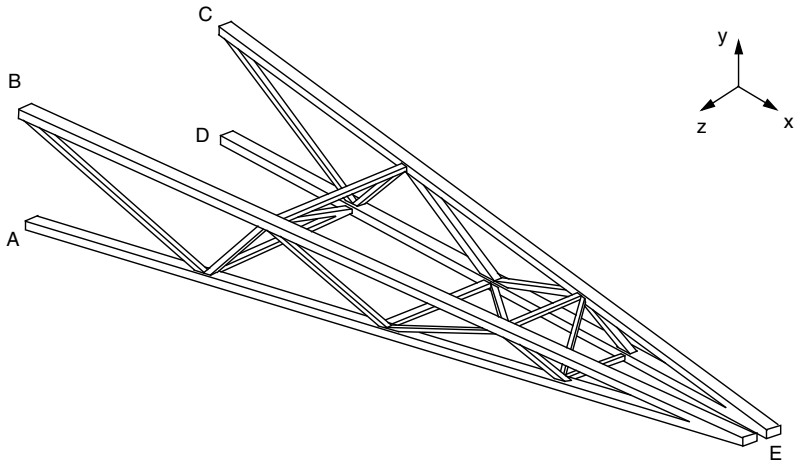


Figure 6–11 Sketch of a light-service cargo crane.

structures is bolted to the truss structures. These connections can transmit little, if any, moment and, therefore, are treated as pinned joints. Both the internal bracing and cross bracing use steel box beams with smaller cross-sections than the main members of the truss structures. The two truss structures are connected at their ends (at point E) in such a way that allows independent movement in the 3-direction and all of the rotations, while constraining the displacements in the 1- and 2-directions to be the same. The crane is welded firmly to a massive structure at points A, B, C, and D. The dimensions of the crane are shown in Figure 6–12. In the following figures, truss A is the structure consisting of members AE, BE, and their internal bracing; and truss B consists of members CE, DE, and their internal bracing.

The ratio of the typical cross-section dimension to global axial length in the main members of the crane is much less than 1/15. The ratio is approximately 1/15 in the shortest member used for internal bracing. Therefore, it is valid to use beam elements to model the crane.

6.4.1 Coordinate system

You should use the default global rectangular Cartesian coordinate system shown in Figure 6–11 and Figure 6–12. Locate the origin of the coordinate system midway between points A and D. If you build your model with a different origin or orientation of the coordinate system, ensure that the input data in your model reflect your coordinate system and not the one shown here.

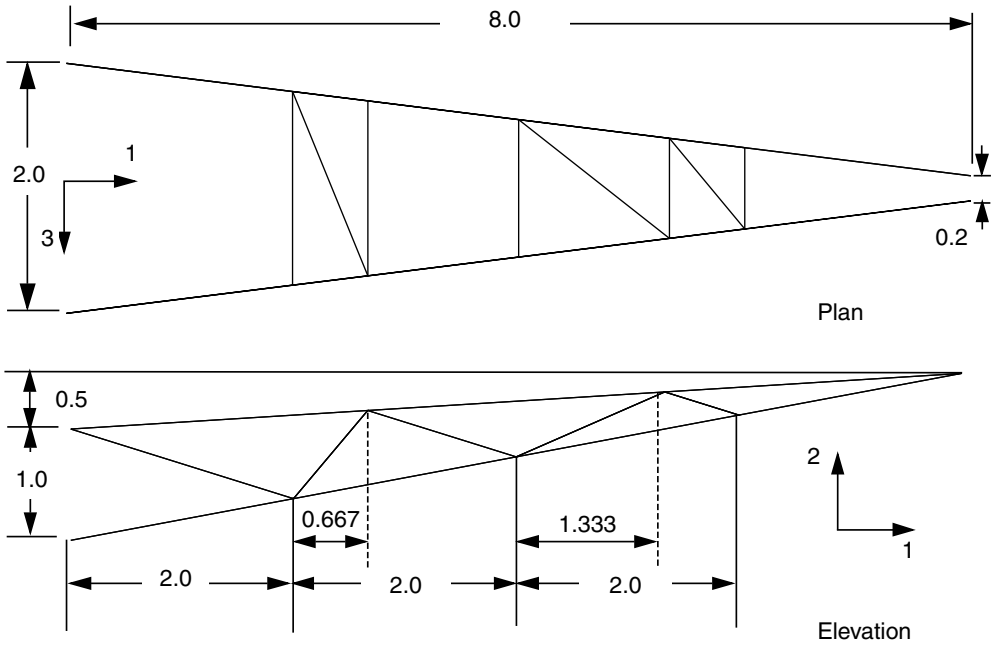


Figure 6–12 Dimensions (in m) of the cargo crane.

6.4.2 Mesh design

The cargo crane will be modeled with three-dimensional, slender, cubic beam elements (B33). The cubic interpolation in these elements allows us to use a single element for each member and still obtain accurate results under the applied bending load. The mesh that you use in the simulation is shown in Figure 6–13.

The welded joints in the crane provide complete continuity of the translations and rotations from one element to the next. You, therefore, need only a single node at each welded joint in the model. The bolted joints, which connect the cross bracing to the truss structures, and the connection at the tip of the truss structures are different. Since these joints do not provide complete continuity for all nodal degrees of freedom, separate nodes are needed for each element at the connection. Appropriate constraints between these separate nodes must then be given by using the ∗MPC, ∗BOUNDARY, or ∗EQUATION options. The ∗MPC and ∗EQUATION options are discussed in more detail later.

The node numbers for the various members of the cargo crane model are shown in Figure 6–14.

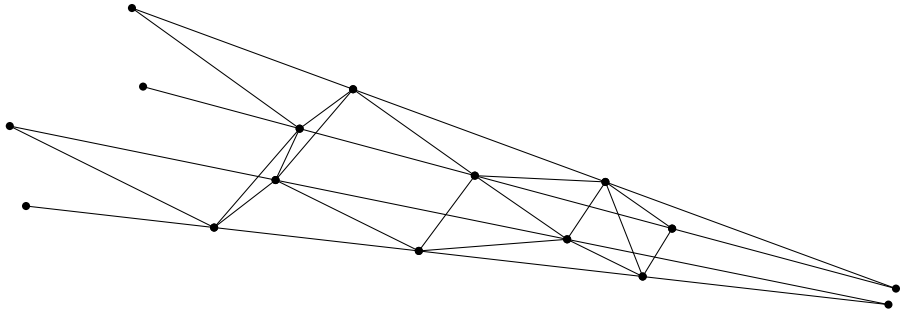


Figure 6–13 Mesh for cargo crane.

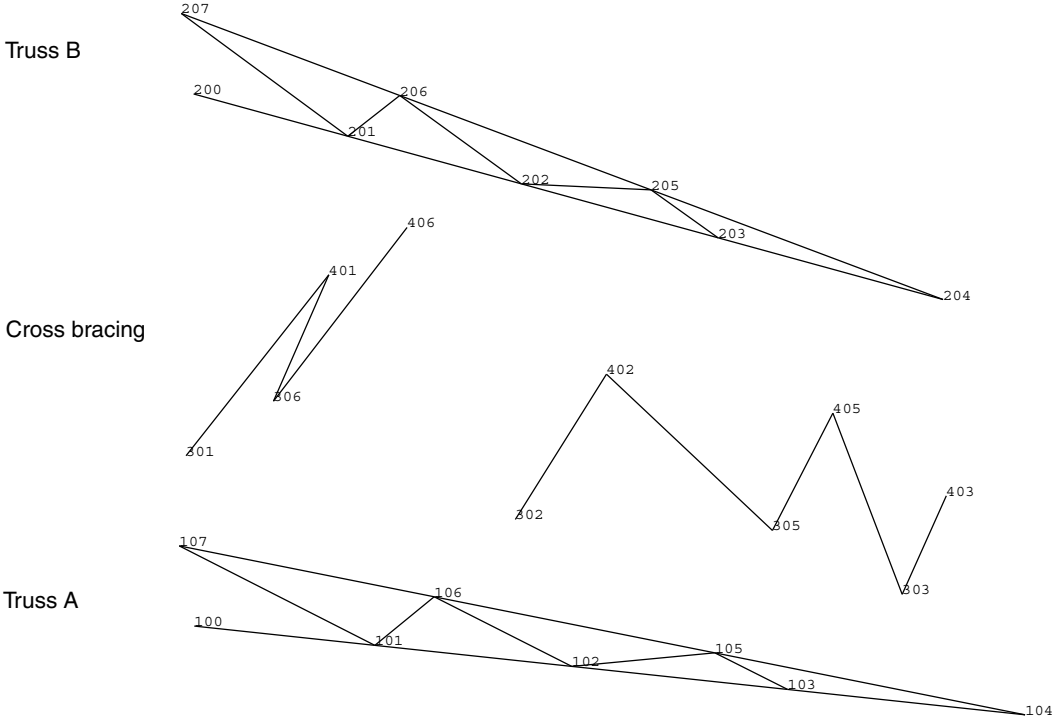


Figure 6–14 Node numbers in crane model.

These are the node numbers from the input file given in “Cargo crane,” Section A.4. Separate nodes have been defined on the cross-bracing elements and the truss structures that they connect. Separate nodes are also needed at the end of each truss structure, point E in Figure 6–11. The node numbers in your model may be different from those shown here.

The element numbers for the various members of the cargo crane model are shown in Figure 6–15.

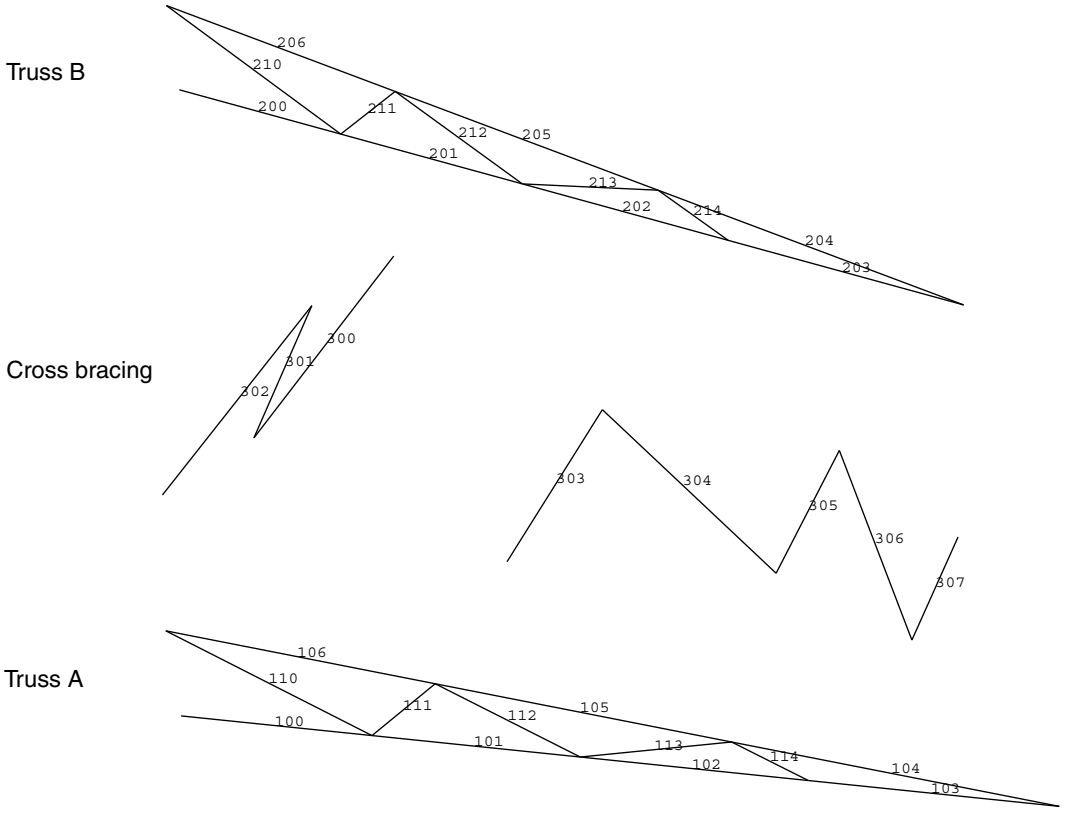


Figure 6–15 Element numbers in crane model.

These are the element numbers from the input file given in “Cargo crane,” Section A.4. The element numbers in your model may be different from those shown here.

6.4.3 Preprocessing—creating the model

The full input file for this example is **crane.inp**, and it is available in "Cargo crane," Section A.4. The Abaqus input options used to create the nodes and elements shown on the preceding pages can be found in "Cargo crane," Section A.4. If you wish to create the entire model using Abaqus/CAE, refer to "Example: cargo crane," Section 6.4 of Getting Started with Abaqus: Interactive Edition.

6.4.4 Reviewing the input file—the model data

This section describes how the model data are described in the input file for this example. These data include the descriptions for the input file heading, its nodes and elements, beam sections and orientations, constraints, and boundary conditions.

Heading

The heading used in this example provides a short description of the model and the units used:

```
*HEADING
3-D model of light-service cargo crane
S.I. Units (m, kg, N, sec)
```

Nodal coordinates and element connectivity

Define the nodal coordinates in a *NODE option block. If you decide to do this with an editor, you may want to use the mesh generation commands found in "Cargo crane," Section A.4. In this example, a node set called **ATTACH** is created; this node set contains the nodes at points A, B, C, and D, the points at which the crane is attached to the parent structure.

```
*NSET, NSET=ATTACH
100, 107, 200, 207
```

Create elements in your model that correspond to the elements shown in Figure 6–15, but remember that your numbering may be different. (Having the same numbering will make defining some modeling features easier, however.) There are several element sets that you will need in this simulation. As the elements are defined, they are grouped into following element sets:

```
OUTA     100, 101, 102, 103, 104, 105, 106
BRACEA   110, 111, 112, 113, 114
OUTB     200, 201, 202, 203, 204, 205, 206
BRACEB   210, 211, 212, 213, 214
CROSSEL  300, 301, 302, 303, 304, 305, 306, 307
```

where these element numbers refer to those elements shown in Figure 6–15. The element sets **OUTA** and **OUTB** contain the main outer members for the two truss structures. Element sets **BRACEA** and

BRACEB contain the elements modeling the internal bracing within each truss structure. Element set **CROSSEL** contains the cross bracing that connects the two truss structures.

Beam element properties

Since the material behavior in this simulation is assumed to be linear elastic, use the *BEAM GENERAL SECTION option to define the section properties. All of the beams in this structure have a box-shaped cross-section.

A box-section is specified using the parameter SECTION=BOX. The first data line contains the section dimensions, which are the dimensions a, b, t_1, t_2, t_3, and t_4 shown in Figure 6–16 for a box section. The dimensions shown in Figure 6–16 are for the main members of the two trusses in the crane.

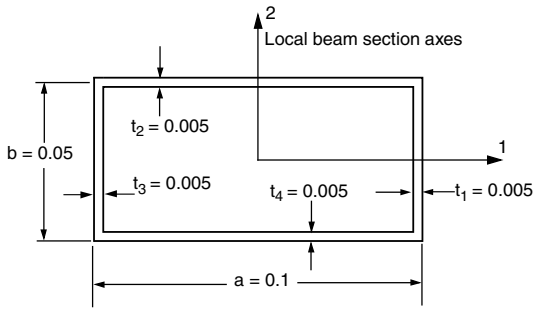


Figure 6–16 Cross-section geometry and dimensions (in m) of the main members.

The beam section axes for the main members should be oriented such that the beam 1-axis is orthogonal to the plane of the truss structures shown in the elevation view (Figure 6–12) and the beam 2-axis is orthogonal to the elements in that plane. Specify this orientation by giving the approximate direction of the beam 1-axis (the $\mathbf{n}_1$-vector) on the second data line of the *BEAM GENERAL SECTION option. To get the correct normal, $\mathbf{n}_2$, in this case, you need to provide a very accurate $\mathbf{n}_1$. It is somewhat easier to provide an approximate direction, which would be the negative 3-direction. However, given the logic that Abaqus uses to determine $\mathbf{n}_2$ given $\mathbf{t}$ and $\mathbf{n}_1$, the normal $\mathbf{n}_2$ is rotated slightly from its proper orientation if we use this approximate $\mathbf{n}_1$. You can specify the same $\mathbf{n}_1$-direction for all elements in each of the two truss structures. The third data line contains the elastic and shear moduli, assuming a mild strength steel with E = 200.0 GPa, ν = 0.25, and G = 80.0 GPa. These modeling data are included in the input file in the following option blocks:

```
*BEAM GENERAL SECTION, SECTION=BOX, ELSET=OUTA
0.10,0.05,0.005,0.005,0.005,0.005
-0.1118, 0.0, -0.9936
200.E9,80.E9
```

```
*BEAM GENERAL SECTION, SECTION=BOX, ELSET=OUTB
0.10,0.05,0.005,0.005,0.005,0.005
-0.1118, 0.0, 0.9936
200.E9,80.E9
```

The dimensions of the beam sections for the bracing members are shown in Figure 6–17. Both the cross bracing and the bracing within each truss structure have the same beam section geometry, but they do not share the same orientation of the beam section axes. Therefore, separate *BEAM GENERAL SECTION options must be used. The bracing is made from the same steel as the main members.

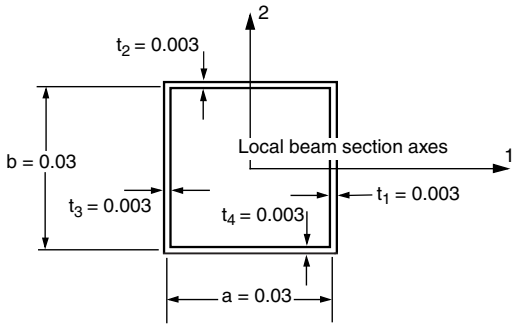


Figure 6–17 Cross-section geometry and dimensions (in m) of the bracing members.

The approximate $\mathbf{n}_1$-vector for the internal truss bracing is the same as for the main members of the respective truss structures. The following input defines the element properties of this bracing:

```
*BEAM GENERAL SECTION, SECTION=BOX, ELSET=BRACEA
0.03,0.03,0.003,0.003,0.003,0.003
-0.1118, 0.0, -0.9936
200.E9,80.E9
*BEAM GENERAL SECTION, SECTION=BOX, ELSET=BRACEB
0.03,0.03,0.003,0.003,0.003,0.003
-0.1118, 0.0, 0.9936
200.E9,80.E9
```

We make some assumptions so that the orientation of the cross bracing is somewhat easier to specify. All of the beam normals ($\mathbf{n}_2$-vectors) should lie approximately in the plane of the plan view of the cargo crane (see Figure 6–12). This plane is skewed slightly from the global 1–3 plane. Again, a simple method for defining such an orientation is to provide an approximate $\mathbf{n}_1$-vector that is orthogonal to this plane on the element property options. The vector should be nearly parallel to the global 2-direction. Since the angle between the normals from one element to the next is always greater than 20°, the normals are not averaged at the nodes.

Depending on the exact orientation of the members in the cross bracing, it is possible that we would have to define the normals individually for each of the cross-bracing elements. Such an exercise would be very similar to what you have already done by defining the normals for the two truss structures. Since the square cross-bracing members are subjected to primarily axial loading, their deformation is not sensitive to cross-section orientation. Therefore, we accept the default normals that Abaqus calculates to be correct. The approximate $\mathbf{n}_1$-vector for the cross bracing is aligned with the y-axis. The following option block specifies the cross-bracing:

```
*BEAM GENERAL SECTION, SECTION=BOX, ELSET=CROSSEL
0.03,0.03,0.003,0.003,0.003,0.003
0.0,1.0,0.0
200.E9,80.E9
```

Beam section orientation

In this model you will have a modeling error if you provide data that only define the orientation of the approximate $\mathbf{n}_1$-vector. The averaging of beam normals at the nodes (see "Beam element curvature," Section 6.1.3) causes Abaqus to use incorrect geometry for the cargo crane model. To see this, you can use Abaqus/Viewer to display the beam section axes and beam tangent vectors (see "Postprocessing," Section 6.4.7). However, the normals in the crane model appear to be correct in Abaqus/Viewer; yet, they are, in fact, slightly incorrect. You can also find such modeling mistakes by examining the averaged nodal normals that are printed in the data (**.dat**) file. Some of the normals in the incorrect model of the cargo crane are shown in the following output:

N O D E D E F I N I T I O N S

NODE NUMBER	COORDINATES			NORMAL			SINGLE POINT CONSTRAINTS TYPE	PLUS	DOF
100	0.00000E+00	0.00000E+00	1.0000	-0.18202	0.98308	2.04813E-02	ENCASTRE		
101	2.0000	0.37500	0.77500	-0.18202	0.98308	2.04813E-02			
102	4.0000	0.75000	0.55000	-0.25486	0.96655	2.86770E-02			
103	6.0000	1.1250	0.32500	-0.18202	0.98308	2.04813E-02			

The problem is the normal vector for node 102, which does not match those at the other nodes defining the lower, main member in truss A (see Figure 6–14). Four elements (101, 102, 112, and 113) contain node 102. When the averaging of beam normals at nodes produces multiple independent normals, the additional normals at the node are also printed in the data file (see "Beam element curvature," Section 6.1.3, for details). The correct geometry for the crane model requires three independent beam normals at node 102: one each for the bracing elements 112 and 113 and a single normal for elements 101 and 102. The normal shown above for node 102 is not the normal needed for elements 101 and 102. If it were, it would match the normals shown for nodes 100, 101, or 103. Nor is it the correct normal for either of the other two elements, whose normals are printed in the data file, as shown below.

N O R M A L D E F I N I T I O N S

ELEMENT	NODE	NORMAL			ELEMENT	NODE	NORMAL		
100	101	-0.1820	0.9831	2.0481E-02	101	101	-0.1820	0.9831	2.0481E-02
101	102	-0.1820	0.9831	2.0481E-02	102	102	-0.1820	0.9831	2.0481E-02
102	103	-0.1820	0.9831	2.0481E-02	103	103	-0.1820	0.9831	2.0481E-02
103	104	-0.1820	0.9831	2.0504E-02	104	105	6.1600E-02	-0.9981	-6.9312E-03

In this table element 113 shows no normal for either of its nodes, node 102 and node 105. Thus, the normal shown for node 102 above in the NODE DEFINITIONS table was the average of the normals from elements 101, 102, and 113. Using the Abaqus logic for averaging normals, we could have predicted that the normals at the nodes of element 113 would be averaged with the normals for the adjacent elements. For this problem the important part of the averaging logic is that normals that subtend an angle less than 20° with the reference normal are averaged with the reference normal to define a new reference normal. In the case of the normals at node 102, the original reference normal is the normal for elements 101 and 102. Since the normal for element 113 at node 102 subtends an angle less than 20° with the original reference normal, it is averaged with the normals for elements 101 and 102 at node 102 to define the new reference normal at that node. On the other hand, since the normal for element 112 subtends an angle of approximately 30° with the original reference normal, it has an independent normal at node 102, as shown in the data file.

This incorrect average normal means that elements 101, 102, and 113 have a section geometry that twists about the beam axis from one end of the element to the other, which is not the intended geometry. You must use the *NORMAL option to define the normals at node 102 for element 113 explicitly. Explicitly specifying the normal directions prevents Abaqus from applying its averaging algorithm. You must also use *NORMAL for the corresponding elements on the opposite side of the crane: element 213, node 202 of truss *B*.

There is also a problem with the normals at nodes 104 and 204 at the tip of each truss structure, again because the angle between elements 103 and 104 is less than 20°. Since we are modeling straight beam elements, the normals are constant at both nodes in each element. Thus, the *NORMAL option block that you should put in your input file has six data lines. If you used the numbering scheme shown in Figure 6–14 and Figure 6–15, the following option block should be added to your input file:

Element

Node

```
*NORMAL, TYPE=ELEMENT
113, 102, -0.3962,  0.9171,  0.0446
113, 105, -0.3962,  0.9171,  0.0446
213, 202, -0.3962,  0.9171, -0.0446
213, 205, -0.3962,  0.9171, -0.0446
103, 104, -0.1820,  0.9829,  0.0205
203, 204, -0.1820,  0.9829, -0.0205
```

Normal at node

Multi-point constraints

The cross bracing, unlike the internal truss bracing, is bolted to the truss members. You can assume that these bolted connections are unable to transmit rotations or torsion. The duplicate nodes that were defined at these locations are needed to define this constraint. In Abaqus such constraints between nodes can be defined using multi-point constraints (MPCs) or constraint equations.

Multi-point constraints allow constraints to be imposed between different degrees of freedom of the model. A large library of MPCs is available in Abaqus. (See "Linear constraint equations," Section 34.2.1 of the Abaqus Analysis User's Manual, for a complete list and a description of each one.) The format of the *MPC option is

```
*MPC
<type of MPC>,<node 1 or node set 1>,<node 2 or node set 2>, ......
```

You can define multiple constraints of the same type with just a single data line by using node sets. The MPC type needed to model the bolted connection is PIN. The pinned joint created by this MPC constrains the displacements at two nodes to be equal, but the rotations, if they exist at the nodes, remain independent.

There are many bolted joints in the crane model. The following is the complete *MPC option block for the model from "Cargo crane," Section A.4:

```
*MPC
PIN,101,301
PIN,102,302
PIN,103,303
PIN,105,305
PIN,106,306
PIN,201,401
PIN,202,402
PIN,203,403
PIN,205,405
PIN,206,406
```

Add a similar option block to your model, changing the node numbers to correspond to those in your model. If all of the nodes on the two truss structures had been grouped into a node set called **TRUSNODE** and all of the nodes on the cross bracing had been grouped into a node set called **CROSNODE**, the option block could have been shortened to the following:

```
*NSET, NSET=TRUSNODE, UNSORTED
101,102,103,105,106,201,202,203,205,206
*NSET, NSET=CROSNODE, UNSORTED
301,302,303,305,306,401,402,403,405,406
*MPC
PIN, TRUSNODE, CROSNODE
```

If a node set is provided as the first item after the MPC type, the second item can be either another node set or a single node. When the data line of an *MPC option contains a node set and then a single node, as shown below, Abaqus creates an MPC constraint between each node in the set and the individual node specified. For example, the following option block would create a pinned joint between node 301 and each node in node set **TRUSNODE**.

```
*MPC
PIN, TRUSNODE, 301
```

Constraint equations

Constraints between nodal degrees of freedom can also be specified with linear equations by using the *EQUATION option. The form of each equation is

$$A_1 u_1 + A_2 u_2 + ... + A_n u_n = 0,$$

where A_i is the coefficient associated with degree of freedom u_i. Each linear constraint equation requires at least two data lines. The number of terms, *n*, involved in the equation is given on the first data line beneath the *EQUATION option. On the subsequent data lines the format is

$$<node_1>,<dof_1>,<A_1>,<node_2>,<dof_2>,<A_2>,\ldots,<node_n>,<dof_n>,<A_n>$$

Exactly four terms must be given on each data line except the final one, which can have fewer terms.

In the crane model the tips of the two trusses are connected together such that degrees of freedom 1 and 2 (the translations in the 1- and 2-directions) of each tip node are equal, while the other degrees of freedom (3–6) at the nodes are independent. We need two linear constraints, one equating degree of freedom 1 at node 104 to degree of freedom 1 at node 204:

$$(1)u_1^{104} + (-1)u_1^{204} = 0,$$

and the other equating degree of freedom 2 at node 104 to degree of freedom 2 at node 204:

$$(1)u_2^{104} + (-1)u_2^{204} = 0.$$

You may have to change the node numbers if you created this model using a preprocessor. The following option block defines the appropriate constraints at point E in the crane model (see Figure 6–11):

```
*EQUATION
2
104,1,1.0, 204,1,-1.0
2
104,2,1.0, 204,2,-1.0
```

The degrees of freedom at the first node defined in an ∗MPC or ∗EQUATION are eliminated from the stiffness matrix. Therefore, these nodes should not appear in other MPCs or constraint equations. Nor should boundary conditions be applied to the eliminated degrees of freedom.

Boundary conditions

The crane is attached firmly to the parent structure. The following ∗BOUNDARY option block constrains all of the nodes at the attachment points, which should have been grouped into node set **ATTACH**:

```
*BOUNDARY
ATTACH, ENCASTRE
```

6.4.5 Reviewing the input file—the history data

The following options specify a static, linear perturbation simulation:

```
*STEP, PERTURBATION
Static tip load on crane
*STATIC
```

Loading

A concentrated load of 10 kN is applied in the negative *y*-direction to node 104. Since there is a constraint equation connecting the *y*-displacement of nodes 104 and 204, the load is carried equally by both nodes. The following ∗CLOAD option block provides an equal load on both nodes:

```
*CLOAD
104,2,-1.0E4
```

Output requests

Write the displacements (U) and reaction forces and moments (RF) at the nodes, and the section forces and moments (SF) in the elements to the output database for postprocessing with Abaqus/Viewer, as shown in the following option blocks:

```
*OUTPUT, FIELD
*NODE OUTPUT
U, RF
*ELEMENT OUTPUT
SF,
*END STEP
```

6.4.6 Running the analysis

Store the input in a file called **`crane.inp`**. Run the analysis using the command

```
abaqus job=crane
```

6.4.7 Postprocessing

Start Abaqus/Viewer by typing the following command at the operating system prompt:

```
abaqus viewer odb=crane
```

Abaqus/Viewer plots the undeformed shape of the crane model.

Plotting the deformed model shape

To begin this exercise, plot the deformed model shape with the undeformed model shape superimposed on it. Specify a nondefault view using (0, 0, 1) as the *X*-, *Y*-, and *Z*-coordinates of the viewpoint vector and (0, 1, 0) as the *X*-, *Y*-, and *Z*-coordinates of the up vector.

Tip: You can also display the model using this view by clicking

from the **Views** toolbar.

The undeformed shape of the crane superimposed upon the deformed shape is shown in Figure 6–18.

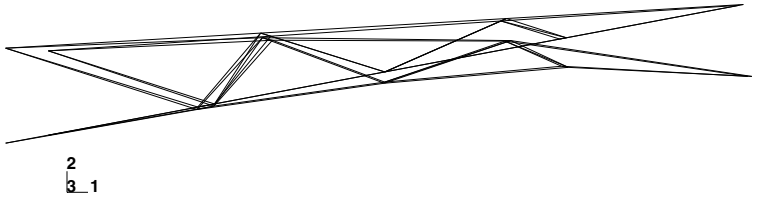


Figure 6–18 Deformed shape of cargo crane.

Using display groups to plot element and node sets

You can use display groups to plot existing node and element sets; you can also create display groups by selecting nodes or elements directly from the viewport. You will create a display group containing only the elements associated with the main members in truss structure A.

To create and plot a display group:

1. In the Results Tree, expand the **Sections** container underneath the output database file named `crane.odb`.
2. To facilitate your selection, change the view back to the default isometric view using the tool in the **Views** toolbar.

   Tip: If the **Views** toolbar is not visible, select **View→Toolbars→Views** from the main menu bar.
3. In succession, click the items in the container until the elements associated with the main members in truss A are highlighted in the viewport. Click mouse button 3 on this item and select **Replace** from the menu that appears.

   Abaqus/Viewer now displays only this group of elements.
4. To save this group, double-click **Display Groups** in the Results Tree; or use the tool in the **Display Group** toolbar.

   The **Create Display Group** dialog box appears.
5. In the **Create Display Group** dialog box, click **Save As** and enter `MainA` as the name for your display group.
6. Click **Dismiss** to close the **Create Display Group** dialog box.

   This display group now appears underneath the **Display Groups** container in the Results Tree.

Beam cross-section orientation

You will now plot the section axes and beam tangents on the undeformed model shape.

To plot the beam section axes:

1. From the main menu bar, select **Plot→Undeformed Shape**; or use the tool in the toolbox to display only the undeformed model shape.
2. From the main menu bar, select **Options→Common**; then, click the **Normals** tab in the dialog box that appears.
3. Toggle on **Show normals**, and accept the default setting of **On elements**.
4. In the **Style** area at the bottom of the **Normals** page, specify the **Length** to be **Long**.
5. Click **OK**.

   The section axes and beam tangents are displayed on the undeformed shape.

The resulting plot is shown in Figure 6–19. The text annotations in Figure 6–19 that identify the section axes and beam tangent will not appear in your image. The vector showing the local beam 1-axis, $\mathbf{n}_1$, is blue; the vector showing the beam 2-axis, $\mathbf{n}_2$, is red; and the vector showing the beam tangent, $\mathbf{t}$, is white.

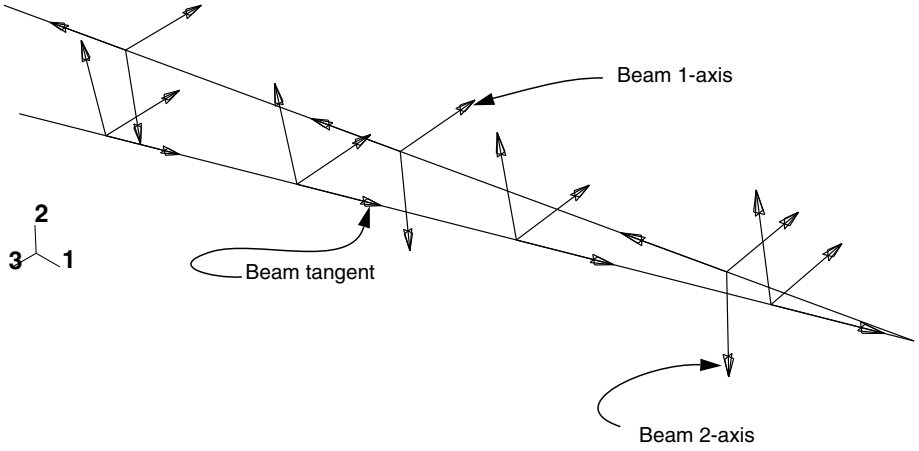


Figure 6–19 Plot of beam section axes and tangents for elements in display group **`MainA`**.

Rendering beam profiles

You will now display an idealized representation of the beam profile and contour the stress results.

To render beam profiles:

1. From the main menu bar, select **View→ODB Display Options**.
   The **ODB Display Options** dialog box appears.
2. In the **General** tabbed page, toggle on **Render beam profiles** and accept the default scale factor of 1.
3. Click **OK**.
   Abaqus/Viewer displays beam profiles with the appropriate dimensions and in the correct orientations. Figure 6–20 shows the beam profiles on the whole model. Your changes are saved for the duration of the session.
4. Click to contour the Mises stress on the rendered profile.

Creating a hard copy

You can save the current image to a file for hardcopy output.

To create a PostScript file of the current image:

1. From the main menu bar, select **File→Print**.
   The **Print** dialog box appears.

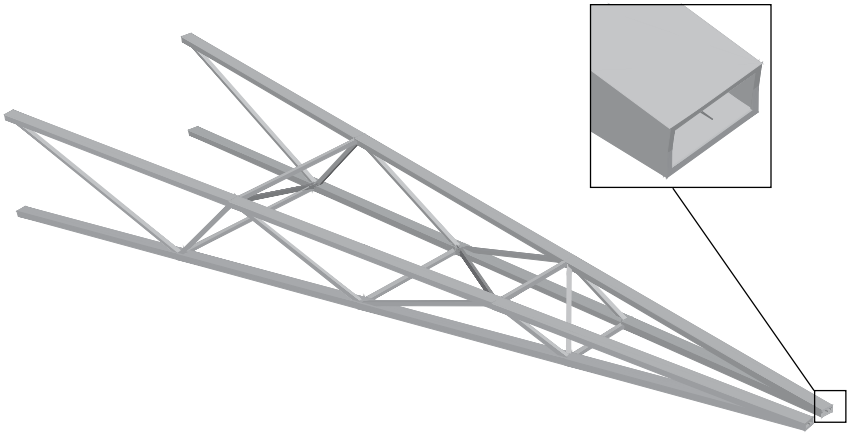


Figure 6–20 Cargo crane with beam profiles displayed.

2. From the **Settings** area in the **Print** dialog box, select **Black&White** as the **Rendition** type; and toggle on **File** as the **Destination**.
3. Select **PS** as the **Format**, and enter **`beam.ps`** as the **File name**.
4. Click .
   The **PostScript Options** dialog box appears.
5. From the **PostScript Options** dialog box, select **600 dpi** as the **Resolution**; and toggle off **Print date**.
6. Click **OK** to apply your selections and to close the dialog box.
7. In the **Print** dialog box, click **OK**.
   Abaqus/Viewer creates a PostScript file of the current image and saves it in your working directory as **`beam.ps`**. You can print this file using your system's command for printing PostScript files.

Displacement summary

Write a summary of the displacements of all nodes in display group **`MainA`** to a file named **`crane.rpt`**. The peak displacement at the tip of the crane in the 2-direction is 0.0188 m.

Section forces and moments

Abaqus can provide output for structural elements in terms of forces and moments acting on the cross-section at a given point. These section forces and moments are defined in the local beam coordinate system. Toggle off the rendering of beam profiles, then contour the section moment

about the beam 1-axis in the elements in display group **MainA**. For clarity, reset the view so that the elements are displayed in the 1–2 plane.

To create a "bending moment"-type contour plot:

1. From the list of variable types on the left side of the **Field Output** toolbar, select **Primary**.
2. From the list of output variables in the center of the toolbar, select **SM**.

   Abaqus/Viewer automatically selects **SM1**, the first component name in the list on the right side of the **Field Output** toolbar, and displays a contour plot of the bending moment about the beam 1-axis on the deformed model shape. The deformation scale factor is chosen automatically since geometric nonlinearity was not considered in the analysis.
3. Open the **Common Plot Options** dialog box, and select a **Uniform** deformation scale factor of **1.0**.

   Color contour plots of this type typically are not very useful for one-dimensional elements such as beams. A more useful plot is a "bending moment"-type plot, which you can produce using the contour options.
4. From the main menu bar, select **Options→Contour**; or use the **Contour Options** tool in the toolbox.

   The **Contour Plot Options** dialog box appears; by default, the **Basic** tab is selected.
5. In the **Contour Type** field, toggle on **Show tick marks for line elements**.
6. Click **OK**.

   The plot shown in Figure 6–21 appears. The magnitude of the variable at each node is now indicated by the position at which the contour curve intersects a "tick mark" drawn perpendicular to the element. This "bending moment"-type plot can be used for any variable (not just bending moments) for any one-dimensional element, including trusses and axisymmetric shells as well as beams.

6.5 Related Abaqus examples

- "Detroit Edison pipe whip experiment," Section 2.1.2 of the Abaqus Example Problems Manual
- "Buckling analysis of beams," Section 1.2.1 of the Abaqus Benchmarks Manual
- "Crash simulation of a motor vehicle," Section 1.3.14 of the Abaqus Benchmarks Manual
- "Geometrically nonlinear analysis of a cantilever beam," Section 2.1.2 of the Abaqus Benchmarks Manual

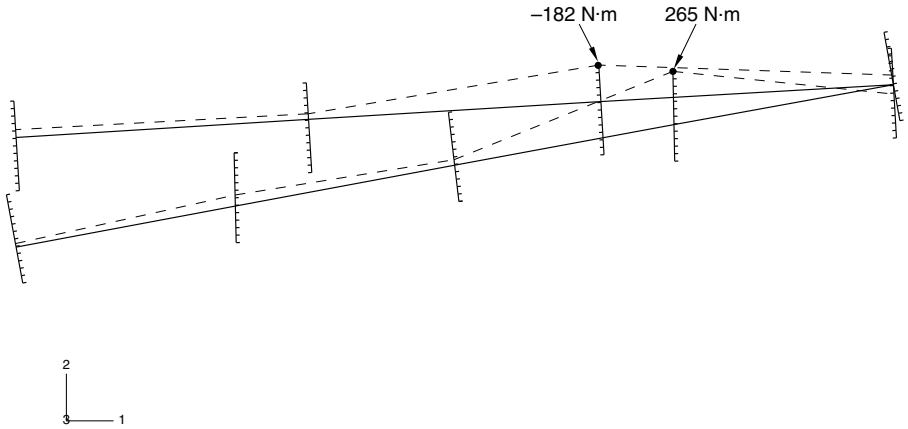


Figure 6–21 Bending moment diagram (moment about beam 1-axis) for elements in display group **`MainA`**. The locations with the highest stress (created by the bending of the elements) are indicated.

6.6 Suggested reading

Basic beam theory

- Timoshenko, S., *Strength of Materials: Part II*, Krieger Publishing Co., 1958.
- Oden, J. T. and E. A. Ripperger, *Mechanics of Elastic Structures*, McGraw-Hill, 1981.

Basic computational beam theory

- Cook, R. D., D. S. Malkus, and M. E. Plesha, *Concepts and Applications of Finite Element Analysis*, John Wiley & Sons, 1989.
- Hughes, T. J. R., *The Finite Element Method*, Prentice-Hall Inc., 1987.

6.7 Summary

- The behavior of beam elements can be determined by numerical integration of the section (either ∗BEAM SECTION or ∗BEAM GENERAL SECTION) or can be given directly in terms of area, moments of inertia, and torsional constant (∗BEAM GENERAL SECTION).
- When ∗BEAM GENERAL SECTION is used with numerical integration, the calculations are done once at the start of the simulation, and elastic behavior is assumed.

- Abaqus includes a number of standard cross-section shapes. Other shapes, provided they are “thin-walled,” can be modeled using SECTION=ARBITRARY.
- The orientation of the cross-section must be defined either by specifying a third node or by defining a vector as part of the element property definition. The orientations can be plotted in Abaqus/Viewer.
- The beam cross-section can be offset from the nodes that define the beam. This procedure is useful in modeling stiffeners on shells.
- The linear and quadratic beams include the effects of shear deformation. The cubic beams in Abaqus/Standard do not account for shear flexibility. The open-section beam elements in Abaqus/Standard correctly model the effects of torsion and warping (including warping constraints) in thin-walled, open sections.
- Multi-point constraints, constraint equations, and connectors can be used to connect degrees of freedom at nodes to model pinned connections, rigid links, etc.
- “Bending moment”-type plots allow the results of one-dimensional elements, such as beams, to be visualized easily.
- Display options allow you to render beam profiles for enhanced graphical representations of undeformed and deformed plots.
- Hard copies of Abaqus/Viewer plots can be obtained in PostScript (PS), Encapsulated PostScript (EPS), Tag Image File Format (TIFF), Portable Network Graphics (PNG), and Scalable Vector Graphics (SVG) formats.

7. Linear Dynamics

A static analysis is sufficient if you are interested in the long-term response of a structure to applied loads. However, if the duration of the applied load is short (such as in an earthquake) or if the loading is dynamic in nature (such as that from rotating machinery), you must perform a dynamic analysis. This chapter discusses linear dynamic analysis in Abaqus/Standard; see Chapter 9, "Nonlinear Explicit Dynamics," for a discussion of nonlinear dynamic analysis in Abaqus/Explicit.

7.1 Introduction

A dynamic simulation is one in which inertia forces are included in the dynamic equation of equilibrium:

$$M\ddot{u} + I - P = 0,$$

where

- M is the mass of the structure,
- $\ddot{u}$ is the acceleration of the structure,
- I are the internal forces in the structure, and
- P are the applied external forces.

The expression in the equation shown above is nothing more than Newton's second law of motion ($F = ma$).

The inclusion of the inertial forces ($M\ddot{u}$) in the equation of equilibrium is the major difference between static and dynamic analyses. Another difference between the two types of simulations is in the definition of the internal forces, I. In a static analysis the internal forces arise only from the deformation of the structure; in a dynamic analysis the internal forces contain contributions created by both the motion (i.e., damping) and the deformation of the structure.

7.1.1 Natural frequencies and mode shapes

The simplest dynamic problem is that of a mass oscillating on a spring, as shown in Figure 7–1.

The internal force in the spring is given by ku so that its dynamic equation of motion is

$$m\ddot{u} + ku - p = 0.$$

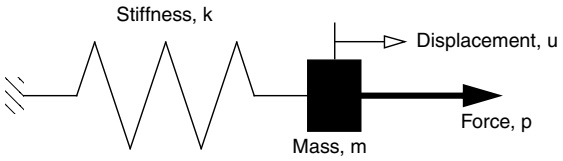


Figure 7–1 Mass-spring system.

This mass-spring system has a *natural frequency* (in radians/time) given by

$$\omega = \sqrt{\frac{k}{m}}.$$

If the mass is moved and then released, it will oscillate at this frequency. If the force is applied at this frequency, the amplitude of the displacement will increase dramatically—a phenomenon known as resonance.

Real structures have a large number of natural frequencies. It is important to design structures in such a way that the frequencies at which they may be loaded are not close to the natural frequencies. The natural frequencies can be determined by considering the dynamic response of the unloaded structure ($P = 0$ in the dynamic equilibrium equation). The equation of motion is then

$$M\ddot{u} + I = 0.$$

For an undamped system $I = Ku$, so

$$M\ddot{u} + Ku = 0.$$

Solutions to this equation have the form

$$u = \phi e^{i\omega t}.$$

Substituting this into the equation of motion yields the *eigenvalue* problem

$$K\phi = \lambda M\phi,$$

where $\lambda = \omega^2$.

This system has n eigenvalues, where n is the number of degrees of freedom in the finite element model. Let λ_j be the *j*th eigenvalue. Its square root, ω_j, is the *natural frequency* of the *j*th mode of the structure, and ϕ_j is the corresponding *j*th *eigenvector*. The eigenvector is also known as the *mode shape* because it is the deformed shape of the structure as it vibrates in the *j*th mode.

In Abaqus the *FREQUENCY procedure is used to extract the modes and frequencies of the structure. This procedure is easy to use in that you need only specify the number of modes required or the maximum frequency of interest.

7.1.2 Modal superposition

The natural frequencies and mode shapes of a structure can be used to characterize its dynamic response to loads in the linear regime. The deformation of the structure can be calculated from a combination of the mode shapes of the structure using the *modal superposition* technique. Each mode shape is multiplied by a scale factor. The vector of displacements in the model, u, is defined as

$$u = \sum_{i=1}^{\infty} \alpha_i \phi_i,$$

where α_i is the modal displacement and ϕ_i is the generalized coordinate of mode i. This technique is valid only for simulations with small displacements, linear elastic materials, and no contact conditions—in other words, linear problems.

In structural dynamic problems the response of a structure usually is dominated by a relatively small number of modes, making modal superposition a particularly efficient method for calculating the response of such systems. Consider a model containing 10,000 degrees of freedom. Direct integration of the dynamic equations of motion would require the solution of 10,000 simultaneous equations at each point in time. If the structural response is characterized by 100 modes, only 100 equations need to be solved every time increment. Moreover, the modal equations are uncoupled, whereas the original equations of motion are coupled. There is an initial cost in calculating the modes and frequencies, but the savings obtained in the calculation of the response greatly outweigh the cost.

If nonlinearities are present in the simulation, the natural frequencies may change significantly during the analysis, and modal superposition cannot be employed. In this case direct integration of the dynamic equation of equilibrium is required, which is much more expensive than modal analysis.

A problem should have the following characteristics for it to be suitable for linear transient dynamic analysis:

- The system should be linear: linear material behavior, no contact conditions, and no nonlinear geometric effects.
- The response should be dominated by relatively few frequencies. As the frequency content of the response increases, such as is the case in shock and impact problems, the modal superposition technique becomes less effective.
- The dominant loading frequencies should be in the range of the extracted frequencies to ensure that the loads can be described accurately.
- The initial accelerations generated by any suddenly applied loads should be described accurately by the eigenmodes.
- The system should not be heavily damped.

7.2 Damping

If an undamped structure is allowed to vibrate freely, the magnitude of the oscillation is constant. In reality, however, energy is dissipated by the structure's motion, and the magnitude of the oscillation decreases until the oscillation stops. This energy dissipation is known as damping. Damping is usually assumed to be viscous or proportional to velocity. The dynamic equilibrium equation can be rewritten to include damping as

$$M\ddot{u} + I - P = 0,$$

$$I = Ku + C\dot{u},$$

where

C is the damping matrix for the structure and

$\dot{u}$ is the velocity of the structure.

The dissipation of energy is caused by a number of effects, including friction at the joints of the structure and localized material hysteresis. Damping is a convenient way of including the important absorption of energy without modeling the effects in detail.

In Abaqus/Standard the eigenmodes are calculated for the undamped system, yet most engineering problems involve some kind of damping, however small. The relationship between the damped natural frequency and the undamped natural frequency for each mode is

$$\omega_d = \omega\sqrt{1 - \xi^2},$$

where

ω_d is the damped eigenvalue,

$\xi = \frac{c}{c_{cr}}$ is the damping ratio, which is the fraction of critical damping,

c is the damping of that mode shape, and

c_{cr} is the critical damping.

The eigenfrequencies of the damped system are very close to the corresponding quantities for the undamped system for small values of ξ ($\xi < 0.1$). As ξ increases, the undamped eigenfrequencies become less accurate; and as ξ approaches 1, the use of undamped eigenfrequencies becomes invalid.

If a structure is critically damped ($\xi = 1$), after any disturbance it will return to its initial static configuration as quickly as possible without overshooting (Figure 7–2).

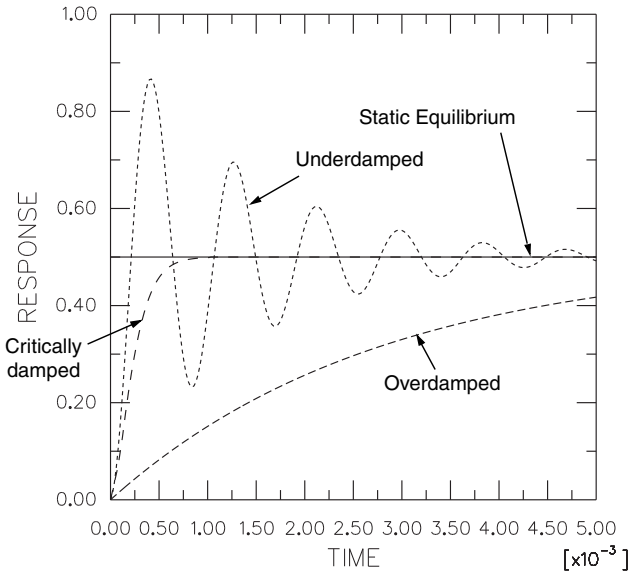


Figure 7–2 Damped motion patterns for various values of ξ.

7.2.1 Definition of damping in Abaqus/Standard

In Abaqus/Standard a number of different types of damping can be defined for a transient modal analysis: direct modal damping, Rayleigh damping, and composite modal damping.

Damping is defined for modal dynamic procedures by using the *MODAL DAMPING option. This option is part of the step definition and allows different amounts of damping to be defined for each mode. Direct, Rayleigh, and composite damping can all be defined this way.

Direct modal damping

The fraction of critical damping, ξ, associated with each mode can be defined using direct modal damping. Typically, values in the range of 1% to 10% of critical damping are used. Direct modal damping allows you to define precisely the damping of each mode of the system.

The MODAL=DIRECT parameter on the *MODAL DAMPING option indicates that direct modal damping is being specified. For example, to define 4% of critical modal damping for the first 10 modes and 5% for modes 11–20, include the following in the step definition:

```
*MODAL DAMPING, MODAL=DIRECT
1, 10, 0.04
11, 20, 0.05
```

Rayleigh damping

In Rayleigh damping the assumption is made that the damping matrix is a linear combination of the mass and stiffness matrices,

$$C = \alpha M + \beta K,$$

where α and β are user-defined constants. Although the assumption that the damping is proportional to the mass and stiffness matrices has no rigorous physical basis, in practice the damping distribution rarely is known in sufficient detail to warrant any other more complicated model. In general, this model ceases to be reliable for heavily damped systems; that is, above approximately 10% of critical damping. As with the other forms of damping, you can define precisely the Rayleigh damping of each mode of the system.

For a given mode i, the damping ratio, ξ_i, and the Rayleigh damping values, α and β, are related through

$$\xi_i = \frac{\alpha}{2\omega_i} + \frac{\beta\omega_i}{2}.$$

The RAYLEIGH parameter on the *MODAL DAMPING option indicates that Rayleigh damping is to be used. For example, to define $\alpha = 0.2525$ and $\beta = 2.9 \times 10^{-3}$ for modes 1–10 and $\alpha = 0.2727$ and $\beta = 3.03 \times 10^{-3}$ for modes 11–20, the following lines would be included in the step definition:

```
*MODAL DAMPING, RAYLEIGH
1, 10, 0.2525, 2.9E-3
11, 20, 0.2727, 3.03E-3
```

Composite damping

In composite damping a fraction of critical damping is defined for each material, and a composite damping value is found for the whole structure. This option is useful when many different materials are present in the structure. Composite damping is not discussed further in this guide.

7.2.2 Choosing damping values

In most linear dynamic problems the proper specification of damping is important to obtain accurate results. However, damping is approximate in the sense that it models the energy absorbing characteristics of the structure without attempting to model the physical mechanisms that cause them. Therefore, it is difficult to determine the damping data required for a simulation. Occasionally, you may have data available from dynamic tests, but often you will have to work with data gleaned from references or experience. In such cases you should be very cautious in interpreting the results, and you should use parametric studies to assess the sensitivity of the simulation to damping values.

7.3 Element selection

Virtually all of the elements in Abaqus can be used in dynamic analyses. In general the rules for selecting the elements are the same as those for static simulations. However, for simulations of impact and blast loading, first-order elements should be used. They have a lumped mass formulation, which is better able to model the effect of stress waves than the consistent mass formulation used in the second-order elements.

7.4 Mesh design for dynamics

When you are designing meshes for dynamic simulations, you need to consider the mode shapes that will be excited in the response and use a mesh that is able to represent those mode shapes adequately. This means that a mesh that is adequate for a static simulation may be unsuitable for calculating the dynamic response to loading that excites high frequency modes.

Consider, for example, the plate shown in Figure 7–3. The mesh of first-order shell elements is adequate for a static analysis of the plate under a uniform load and is also suitable for the prediction of the first mode shape. However, the mesh is clearly too coarse to be able to model the sixth mode accurately.

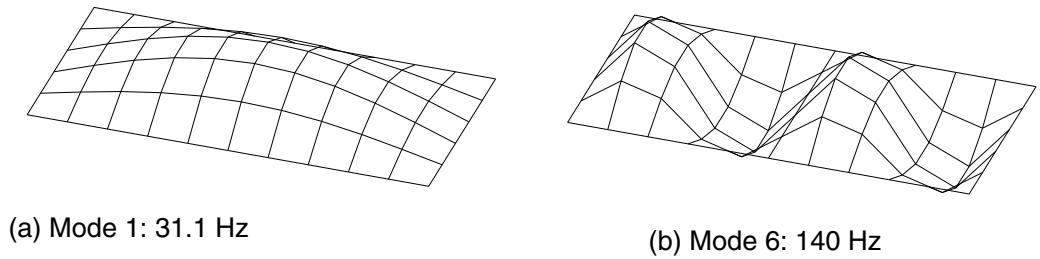
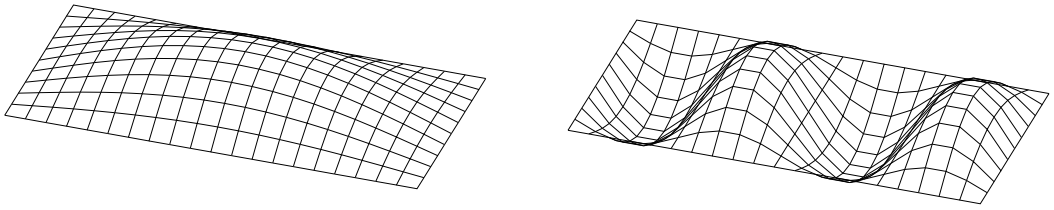


Figure 7–3 Vibration frequencies and corresponding mode shapes of the plate based on the coarse mesh.

Figure 7–4 shows the same plate modeled with a refined mesh of first-order elements. The displaced shape for the sixth mode now looks much better, and the frequency predicted for this mode is more accurate. If the dynamic loading on the plate is such that there is significant excitation of this mode, the refined mesh must be used; the results from the coarse mesh will not be accurate.



(a) Mode 1: 30.2 Hz (b) Mode 6: 124 Hz

Figure 7–4 Vibration frequencies and corresponding mode shapes of the plate based on the fine mesh.

7.5 Example: cargo crane under dynamic loading

This example uses the same cargo crane that you analyzed in "Example: cargo crane," Section 6.4, but you have now been asked to investigate what happens when a load of 10 kN is dropped onto the lifting hook for 0.2 seconds. The connections at points *A*, *B*, *C*, and *D* (see Figure 7–5) can only withstand a maximum pull-out force of 100 kN. You have to decide whether or not any of these connections will break.

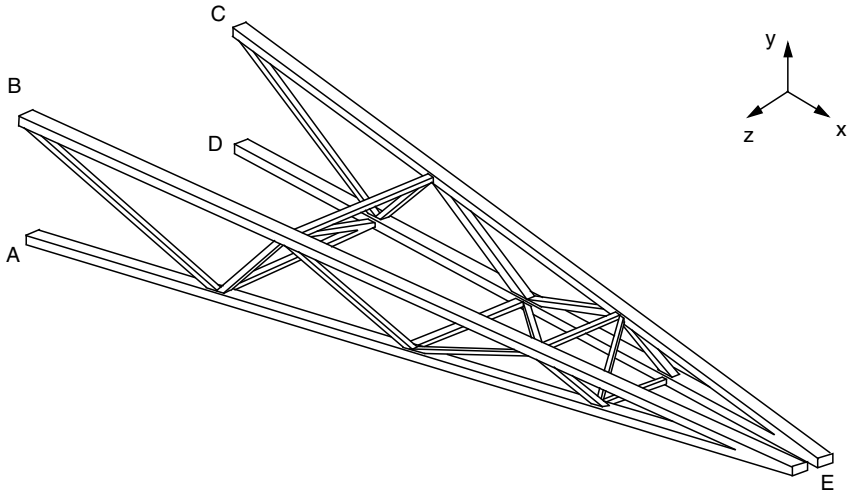


Figure 7–5 Cargo crane.

The short duration of the loading means that inertia effects are likely to be important, making dynamic analysis essential. You are not given any information regarding the damping of the structure. Since there are bolted connections between the trusses and the cross bracing, the energy absorption caused by frictional effects is likely to be significant. Based on experience, you therefore choose 5% of critical damping in each mode.

The magnitude of the applied load versus time is shown in Figure 7–6.

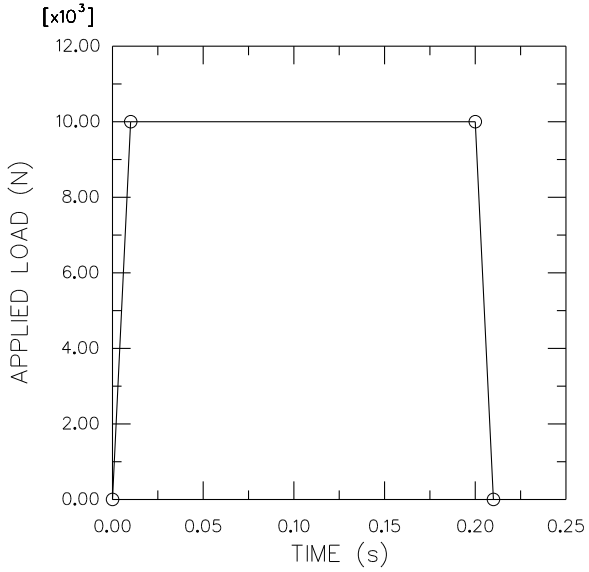


Figure 7–6 Load-time characteristic.

The steps that follow assumes that you have access to the full input file for this example. This input file, **`dynamics.inp`**, is provided in "Cargo crane – dynamic loading," Section A.5, in the online HTML version of this manual. Instructions on how to fetch and run the script are given in Appendix A, "Example Files."

If you prefer to create this example interactively using Abaqus/CAE, refer to "Example: cargo crane under dynamic loading," Section 7.5 of Getting Started with Abaqus: Interactive Edition.

7.5.1 Modifications to the input file—the model data

The model data are the same as for the static analysis with the modifications described below. These modifications are most easily made using an editor, although you may change the model in a preprocessor if you prefer.

Material

In dynamic simulations the density of every material must be specified so that the mass matrix can be formed. The steel in the crane has a density of 7800 kg/m^3. The beam element properties were defined using the *BEAM GENERAL SECTION option, so there are no material property definitions in this input file. The density must be specified using the DENSITY parameter on the *BEAM GENERAL SECTION option. For example,

```
*BEAM GENERAL SECTION, SECTION=BOX, ELSET=OUTA, DENSITY=7800.
0.10,0.05,0.005,0.005,0.005,0.005
-0.1118, 0.0, -0.9936
200.E9,80.E9
```

The DENSITY parameter has been added to all the element property options.

If material data are defined using the *MATERIAL option, the density is included by using the *DENSITY option and giving the mass density on the data line. For example,

```
*MATERIAL, NAME=STEEL
*ELASTIC
<Young's modulus>,<Poisson's ratio>
*DENSITY
<density>,
```

Initial conditions

In this example the structure has no initial velocities or accelerations, which is the default. However, if you wanted to define initial velocities, you could do so using the following option:

```
*INITIAL CONDITIONS, TYPE=VELOCITY
```

The nodes (or node sets), the direction, and the magnitude of the velocity are specified on the data line, as follows:

<node or node set>,<dof>, <velocity>

For example:

```
*INITIAL CONDITIONS, TYPE=VELOCITY
NALL, 1, 10.0
```

would set the velocity in the 1-direction of all the nodes in node set **NALL** to 10 m/s.

Time variation of load

The magnitude of the load applied to the tip of the crane is time dependent as illustrated in Figure 7–6. The time dependence of a load is defined using the *AMPLITUDE option. The *AMPLITUDE option must appear as part of the model data, even though the *CLOAD option referring to it is part of the history data.

Four pairs of time and magnitude data are given on each data line for the *AMPLITUDE option, and a name is assigned to the amplitude curve using the NAME parameter. For your simulation the option block defining the amplitude curve should look similar to the following:

```
*AMPLITUDE, NAME=BOUNCE, VALUE=RELATIVE, SMOOTH=0.25
0.0, 0.0, 0.01, 1.0, 0.2, 1.0, 0.21, 0.0
```

The name of the curve, **BOUNCE**, will be used to refer the loading option (*CLOAD) to this amplitude curve. The actual load applied will be the product of the magnitude on the loading option and the amplitude on the **BOUNCE** curve. The parameter VALUE=RELATIVE is used to indicate this approach. You can choose to define the absolute magnitude of the loading on the *AMPLITUDE option by using VALUE=ABSOLUTE.

7.5.2 Modifications to the input file—the history data

The history definition is substantially different from that in the static analysis. Therefore, delete the entire static step, and add a new history section as discussed below.

Two steps are required for this analysis. The first step calculates the natural frequencies and mode shapes of the structure. The second step then uses these data to calculate the transient dynamic response of the hoist. If you want to model any nonlinearities in this simulation, you must use the *DYNAMIC procedure. In this analysis we will assume that everything is linear.

Step 1 – Modes and frequencies

The *FREQUENCY procedure is used to calculate natural frequencies and mode shapes. Abaqus offers the Lanczos and the subspace iteration eigenvalue extraction methods. The Lanczos method is the default method; it is generally faster when a large number of eigenmodes is required for a system with many degrees of freedom. The subspace iteration method may be faster when only a few (less than 20) eigenmodes are needed.

We use the default Lanczos eigensolver in this analysis. The number of modes required is specified on the data line of the *FREQUENCY option. Alternatively, it is possible to specify the minimum and maximum frequencies of interest, so that the step will complete once Abaqus has found all of the eigenvalues inside the specified range. A shift point may also be specified so that eigenvalues nearest the shift point will be extracted. By default, no minimum or maximum frequency or shift is used. If the structure is not constrained against rigid body modes, the shift value should be set to a small negative value to remove numerical problems associated with rigid body motion.

The form of the *FREQUENCY option block is

```
*FREQUENCY
<number of eigenvalues>,< min. frequency>,< max. frequency>,<shift point>
```

The step and procedure option blocks for this simulation are

```
*STEP, PERTURBATION
Frequency extraction of the first 30 modes
```

```
*FREQUENCY
30,
```

In structural dynamic analysis the response is usually associated with the lower modes. However, enough modes should be extracted to provide a good representation of the dynamic response of the structure. One way of checking that a sufficient number of eigenvalues has been extracted is to look at the total effective mass in each degree of freedom, which indicates how much of the mass is active in each direction of the extracted modes. The effective masses are tabulated in the data file under the eigenvalue output. Ideally, the sum of the modal effective masses for each mode in each direction should be at least 90% of the total mass. This is discussed further in "Effect of the number of modes," Section 7.6.

Boundary conditions

The boundary conditions are the same as in the static analysis.

Output

By default, Abaqus writes the mode shapes to the output database (**.odb**) file so that they can be plotted using Abaqus/Viewer. The nodal displacements for each mode shape are normalized so that the maximum displacement is unity. Therefore, these results, and the corresponding stresses and strains, are not physically meaningful: they should be used only for relative comparisons.

The step terminates with

```
*END STEP
```

Step 2 – Transient dynamics

The *MODAL DYNAMIC procedure is used for transient modal dynamic analysis. The fixed time increment and the total step time are given on the data line for this option. The total time of the simulation is 0.5 seconds with a constant increment of 0.005 seconds. The format of this data line is basically the same as that for *STATIC. However, in this case we must be careful to ensure that we give real values of time; in dynamic analysis time is a real, physical quantity.

The form of the *STEP and *MODAL DYNAMIC option blocks for this simulation should be

```
*STEP, PERTURBATION
Crane Response to Dropped Load
*MODAL DYNAMIC
0.005, 0.5
```

Damping

5% of critical damping should be used in all 30 modes extracted in the first step. This input is specified in the following *MODAL DAMPING option block:

```
*MODAL DAMPING, MODAL=DIRECT
1, 30, 0.05
```

Selecting the eigenmodes

The eigenmodes used in a mode-based dynamic procedure must be selected with the *SELECT EIGENMODES option if *MODAL DAMPING is used. For this example, the form of this option is:

```
*SELECT EIGENMODES, GENERATE
1, 30, 1
```

Loading

Apply the concentrated force to the tip of the crane at node 104 in the negative global 2-direction. The *EQUATION constraint between nodes 104 and 204 in degree of freedom 2 means that the load will be carried equally by both nodes and, thus, by both halves of the crane. The concentrated force is defined using the *CLOAD option. This example uses the parameter AMPLITUDE=BOUNCE to indicate that the amplitude curve named **BOUNCE** (previously defined as part of the model data) should be used to define the time varying magnitude of the load during the step:

```
*CLOAD, AMPLITUDE=BOUNCE
104, 2, -1.0E4
```

The actual magnitude of the load applied at any point in time is obtained by multiplying the magnitude given on the *CLOAD option (−10,000 N) and the value of the **BOUNCE** amplitude curve at that time.

Boundary conditions

The same boundary conditions that were applied in Step 1 are still in effect for this step. Since the boundary conditions cannot be changed between a *FREQUENCY step and any subsequent modal dynamic steps, no boundary conditions should be specified.

Output

Dynamic analyses usually require many more increments than static analyses to complete. As a consequence, the volume of output from dynamic analyses can be very large, and you should control the output requests to keep the output files to a reasonable size.

You can estimate the size of the restart file using the approximate sizes given near the bottom of the data file during a **datacheck** analysis.

In this example request output of the deformed shape to the output database file at the end of every fifth increment. There will be 100 increments in the step (0.5/0.005); therefore, there will be 20 frames of output.

```
*OUTPUT, FIELD, FREQUENCY=5, VARIABLE=PRESELECT
```

The displacements of the independent tip node, which is assigned to a node set named **TIP**, and the reaction forces at the fixed nodes, which are grouped into a node set named **ATTACH**, are written as history data to the output database file every increment so that a higher resolution of these

data will be available. In dynamic analyses we are also concerned about the energy distribution in the model and what form the energy takes. Kinetic energy is present in the model as a result of the motion of the mass; strain energy is present as a result of the displacement of the structure; energy is also dissipated through damping. We can output the kinetic energy (ALLKE), strain energy (ALLSE), energy dissipated through damping (ALLVD), external work on the entire model (ALLWK), and the total energy balance in the model (ETOTAL). The history portion of the output request is written as follows:

```
*NSET, NSET=TIP
104,
*OUTPUT, HISTORY, FREQUENCY=1
*NODE OUTPUT, NSET=TIP
U,
*NODE OUTPUT, NSET=ATTACH
RF,
*ENERGY OUTPUT
ALLKE, ALLSE, ALLVD, ALLWK, ETOTAL
```

The step terminates with the following:

```
*END STEP
```

7.5.3 Running the analysis

The input file is called **dynamics.inp** (an example is listed in "Cargo crane – dynamic loading," Section A.5). Use the following command to run the analysis in the background:

```
abaqus job=dynamics
```

7.5.4 Results

Examine the status (**.sta**) file and printed output data (**.dat**) file to evaluate the analysis results.

Status file

Looking at the contents of the status file, **dynamics.sta**, we can see that the time increment associated with the single increment in Step 1 is very small. A *FREQUENCY step uses no time, because time is not relevant in a frequency extraction step. The contents of the status file are shown below.

```
SUMMARY OF JOB INFORMATION:
STEP   INC ATT SEVERE EQUIL TOTAL  TOTAL      STEP       INC OF       DOF    IF
               DISCON ITERS ITERS  TIME/    TIME/LPF    TIME/LPF    MONITOR RIKS
               ITERS               FREQ
   1     1   1     0     0     0  0.000      1.00e-036  1.000e-036
   2     1   1     0     0     0  0.000      0.00500    0.005000
   2     2   1     0     0     0  0.000      0.0100     0.005000
   2     3   1     0     0     0  0.000      0.0150     0.005000
```

```
   2     4   1     0     0     0  0.000      0.0200     0.005000
   2     5   1     0     0     0  0.000      0.0250     0.005000
   2     6   1     0     0     0  0.000      0.0300     0.005000
....
   2    94   1     0     0     0  0.000      0.470      0.005000
   2    95   1     0     0     0  0.000      0.475      0.005000
   2    96   1     0     0     0  0.000      0.480      0.005000
   2    97   1     0     0     0  0.000      0.485      0.005000
   2    98   1     0     0     0  0.000      0.490      0.005000
   2    99   1     0     0     0  0.000      0.495      0.005000
   2   100   1     0     0     0  0.000      0.500      0.005000
```

The output in the status file for Step 2 shows that the time increment size is constant throughout the step and that each increment requires only one iteration. Since modal dynamic analysis involves the linear superposition of the mode shapes, no iterating is required. For the same reason, the message file contains no information about equilibrium or residuals.

Data file

The primary results for Step 1 are the extracted eigenvalues, participation factors, and effective mass, as shown below.

E I G E N V A L U E O U T P U T

MODE NO	EIGENVALUE	FREQUENCY (RAD/TIME)	FREQUENCY (CYCLES/TIME)	GENERALIZED MASS	COMPOSITE MODAL DAMPING
1	1773.4	42.112	6.7023	151.92	0.0000
2	7016.8	83.766	13.332	30.206	0.0000
3	7644.1	87.430	13.915	90.401	0.0000
4	22999.	151.65	24.136	250.63	0.0000
5	24714.	157.21	25.020	275.90	0.0000
6	34811.	186.58	29.695	493.16	0.0000
7	42748.	206.76	32.906	1107.1	0.0000
....					
25	2.26885E+05	476.32	75.809	207.47	0.0000
26	2.42800E+05	492.75	78.423	127.02	0.0000
27	2.84057E+05	532.97	84.825	1240.8	0.0000
28	2.92452E+05	540.79	86.069	330.69	0.0000
29	3.13943E+05	560.31	89.175	272.41	0.0000
30	3.64774E+05	603.96	96.124	64.980	0.0000

The highest frequency extracted is 96 Hz. The period associated with this frequency is 0.0104 seconds, which is comparable to the fixed time increment of 0.005 seconds. There is no point in extracting modes whose period is substantially smaller than the time increment used. Conversely, the time increment must be capable of resolving the highest frequencies of interest.

The column for generalized mass lists the mass of a single degree of freedom system associated with that mode.

The table of participation factors indicates the predominant degrees of freedom in which the modes act, as shown below.

P A R T I C I P A T I O N F A C T O R S

MODE NO	X-COMPONENT	Y-COMPONENT	Z-COMPONENT	X-ROTATION	Y-ROTATION	Z-ROTATION
1	-6.11690E-04	-6.14531E-03	1.4284	1.4276	-6.0252	-3.34721E-02
2	0.18470	-0.25678	8.31883E-04	2.09977E-03	-6.05062E-03	-1.7751
3	-0.17440	1.5515	4.88139E-03	-5.59953E-03	3.24483E-02	9.3618
4	-8.69482E-05	-9.61288E-03	8.23644E-02	0.25721	1.2335	-2.97485E-02

```
       5     -3.80669E-03    1.13896E-03   -3.04330E-02   -0.60741         1.7592        -2.01080E-02
       6      3.71619E-02   -0.35674        6.05207E-03   -1.37690E-02    6.71471E-03   -0.98290
       7     -2.48375E-03   -1.58340E-03    6.19483E-02    8.18701E-02   -0.29885         5.73966E-04
....
      25     -8.25367E-02   -0.22220       -3.54513E-02    1.61838E-02   -2.18156E-02   -0.14563
      26     -1.98899E-02   -0.35108        4.61344E-02    1.80975E-03   -1.27588E-02   -0.17942
      27      1.71767E-02    2.51338E-02    2.26535E-02    1.06724E-03   -4.31578E-02    1.92991E-02
      28      4.73374E-02    2.79248E-02   -0.11861       -7.32078E-03    0.24177       -2.37795E-02
      29      9.83570E-03   -3.64867E-03    4.65417E-03   -8.45350E-04   -1.56687E-02   -7.74623E-03
      30      4.83653E-02    1.85437E-02    0.13423        4.49321E-02   -0.35873       -4.29368E-02
```

Mode 1 acts predominately in the 3–direction.

The table of effective mass indicates the amount of mass active in each degree of freedom for any one mode, as shown below. The total mass of the model is given earlier in the data file and is 414.34 kg.

```
                    E F F E C T I V E   M A S S

 MODE NO     X-COMPONENT     Y-COMPONENT     Z-COMPONENT     X-ROTATION      Y-ROTATION      Z-ROTATION

       1      5.68446E-05     5.73740E-03      309.98          309.61          5515.3          0.17021
       2       1.0304          1.9917         2.09036E-05     1.33181E-04     1.10585E-03      95.175
       3       2.7495          217.62         2.15407E-03     2.83449E-03     9.51822E-02      7923.1
       4      1.89478E-06     2.31603E-02      1.7003          16.582          381.34          0.22180
       5      3.99800E-03     3.57904E-04     0.25553          101.79          853.85          0.11155
       6      0.68105          62.760         1.80631E-02     9.34950E-02     2.22352E-02      476.44
       7      6.82949E-03     2.77557E-03      4.2485          7.4203          98.873          3.64709E-04
....
      25       1.4134          10.244         0.26075         5.43401E-02     9.87406E-02      4.4000
      26      5.02489E-02      15.656         0.27034         4.16008E-04     2.06766E-02      4.0887
      27      0.36609         0.78385         0.63677         1.41330E-03      2.3112         0.46216
      28      0.74103         0.25787          4.6522         1.77231E-02      19.329         0.18700
      29      2.63530E-02     3.62651E-03     5.90071E-03     1.94668E-04     6.68780E-02     1.63456E-02
      30      0.15200         2.23444E-02      1.1708         0.13119          8.3622         0.11979

 TOTAL         22.198          378.26          373.68          558.02          8348.4          8695.0
```

To ensure that enough modes have been used, the total effective mass in each direction should be a large proportion of the mass of the model (say 90%). However, some of the mass of the model is associated with nodes that are constrained. This constrained mass is approximately one-quarter of the mass of all the elements attached to the constrained nodes, which, in this case, is approximately 28 kg. Therefore, the mass of the model that is able to move is 385 kg. The effective mass in the *x*-, *y*-, and *z*-directions is 6%, 98%, and 97%, respectively, of the mass that can move. The total effective mass in the 2- and 3-directions is well above the 90% recommended earlier; the total effective mass in the 1-direction is much lower. However, since the loading is applied in the 2-direction, the response in the 1-direction is not significant.

The data file does not contain any results for the modal dynamics step, because all of the data file output requests were turned off.

7.5.5 Postprocessing

When you are in the directory containing the output database file **`dynamics.odb`**, type the following command at the operating system prompt:

```
abaqus viewer odb=dynamics
```

Plotting mode shapes

You can visualize the deformation mode associated with a given natural frequency by plotting the mode shape associated with that frequency.

To select a mode and plot the corresponding mode shape:

1. In the context bar, click the frame selector tool .

   The **Frame Selector** dialog box appears. Drag the bottom corner of the dialog box to enlarge it so that both step names are clearly visible.
2. Drag the frame slider to select frame `1` in **Step-1**. This is the first eigenmode.
3. From the main menu bar, select **Plot→Deformed Shape**; or use the tool in the toolbox.

   Abaqus/Viewer displays the deformed model shape associated with the first vibration mode, as shown in Figure 7–7.

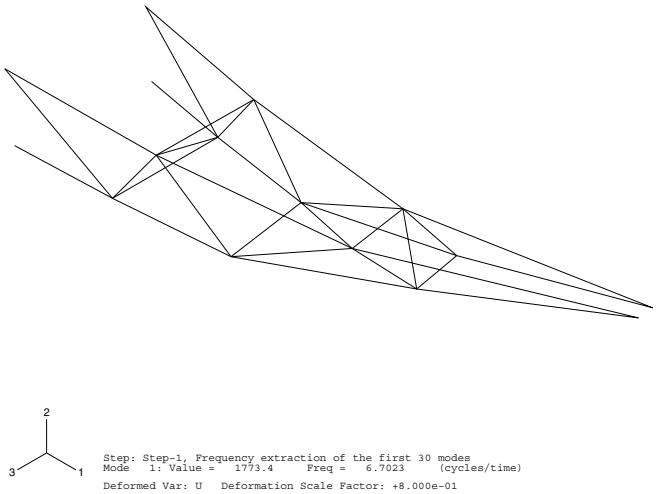

2
3
1
Step: Step-1, Frequency extraction of the first 30 modes
Mode 1: Value = 1773.4 Freq = 6.7023 (cycles/time)
Deformed Var: U Deformation Scale Factor: +8.000e-01

Figure 7–7 Mode 1.

4. Select the third mode (frame `3` in **Step-1**) from the **Frame Selector** dialog box. Afterward, close the dialog box.

   Abaqus/Viewer displays the third mode shape shown in Figure 7–8.

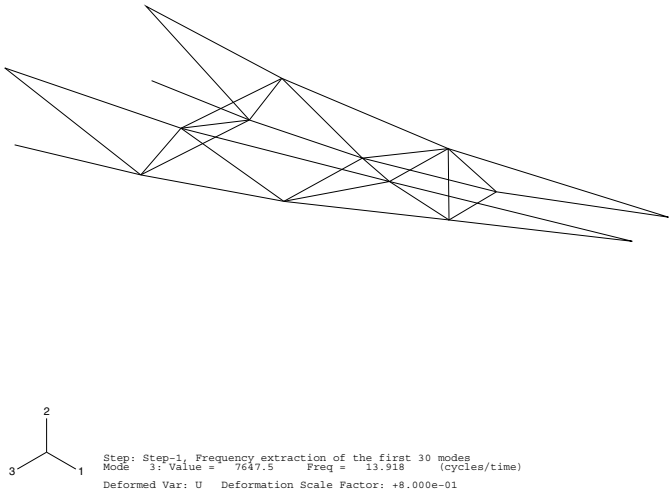

2
3
1
Step: Step-1, Frequency extraction of the first 30 modes
Mode 3: Value = 7647.5 Freq = 13.918 (cycles/time)
Deformed Var: U Deformation Scale Factor: +8.000e-01

Figure 7–8 Mode 3.

Note: A complete list of the available frames is given in the **Step/Frame** dialog box (**Result→Step/Frame**). This dialog box offers an alternative means to switching between frames.

Animation of results

You will animate the analysis results. First create a scale factor animation of the third eigenmode. Then create a time-history animation of the transient results.

To create a scale factor animation of an eigenmode:

1. From the main menu bar, select **Animate→Scale Factor**; or use the tool in the toolbox.

   Abaqus/Viewer displays the third mode shape and steps through different deformation scale factors ranging from 0 to 1.

   Abaqus/Viewer also displays the movie player controls on the right side of the context bar.

2. In the context bar, click to pause the animation.

To create a time-history animation of the transient results:

1. From the main menu bar, select **Result→Active Steps/Frames** to select which frames will be active in the history animation.

   Abaqus/Viewer displays the **Active Steps/Frames** dialog box.
2. Toggle the step names so that only the second step (**Step-2**) is selected.
3. Click **OK** to accept the selection and to close the dialog box.
4. From the main menu bar, select **Animate→Time History**; or use the tool from the toolbox.

   Abaqus/Viewer steps through each available frame of the second step. The state block indicates the current step and increment throughout the animation. After the last increment of this step is reached, the animation process repeats itself.
5. You can customize the deformed shape plot while the animation is running.
   a. Display the **Common Plot Options** dialog box.
   b. Choose **Uniform** from the **Deformation Scale Factor** field.
   c. Enter `15.0` as the deformation scale factor value.
   d. Click **Apply** to apply your change.

      Abaqus/Viewer now steps through the frames in the second step with a deformation scale factor of `15.0`.
   e. Choose **Auto-compute** from the **Deformation Scale Factor** field.
   f. Click **OK** to apply your change and to close the **Common Plot Options** dialog box.

      Abaqus/Viewer now steps through the frames in the second step with a default deformation scale factor of `0.8`.

Determining the peak pull-out force

To find the peak pull-out force at the attachment points, create an *X*–*Y* plot of the reaction force in the 1-direction (variable RF1) at the attached nodes. This involves plotting multiple curves at the same time.

To plot multiple curves:

1. In the Results Tree, click mouse button 3 on **History Output** for the output database named `dynamics.odb`. From the menu that appears, select **Filter**.
2. In the filter field, enter `*RF1*` to restrict the history output to just the reaction force components in the 1-direction.
3. From the list of available history output, select the four curves (using [Ctrl]+Click) that have the following form:

   `Reaction Force:  RF1 PI: TRUSS-1 Node xxx in NSET ATTACH`

4. Click mouse button 3, and select **Plot** from the menu that appears.
   Abaqus/Viewer displays the selected curves.
5. Click in the prompt area to cancel the current procedure.

To position the grid:

1. Double-click the plot to open the **Chart Options** dialog box.
2. In this dialog box, switch to the **Grid Area** tabbed page.
3. In the **Size** region of this page, select the **Square** option.
4. Use the slider to set the size to **75**.
5. In the **Position** region of this page, select the **Auto-align** option.
6. From the available alignment options, select the last one (position the grid in the lower right corner of the viewport).
7. Click **Dismiss**.

To position the legend:

1. Double-click the legend to open the **Chart Legend Options** dialog box.
2. In this dialog box, switch to the **Area** tabbed page.
3. In the **Position** region of this page, toggle on **Inset**.
4. To display the minimum and maximum values in the legend, switch to the **Contents** tabbed page of the dialog box. In the **Numbers** region of this page, toggle on **Show min/max**.
5. Click **Dismiss**.
6. Drag the legend in the viewport to reposition it.

The resulting plot (which has been customized) is shown in Figure 7–9. The curves for the two nodes at the top of each truss (points B and C) are almost a reflection of those for the nodes on the bottom of each truss (points A and D).

Note: To modify the curve styles, click in the Visualization toolbox to open the **Curve Options** dialog box.

At the attachment points at the top of each truss structure, the peak tensile force is around 80 kN, which is below the 100 kN capacity of the connection. Keep in mind that a negative reaction force in the 1-direction means that the member is being pulled away from the wall. The lower attachments are in compression (positive reaction force) while the load is applied but oscillate between tension and compression after the load has been removed. The peak tensile force is about 40 kN, well below the allowable value. To find this value, probe the *X*–*Y* plot.

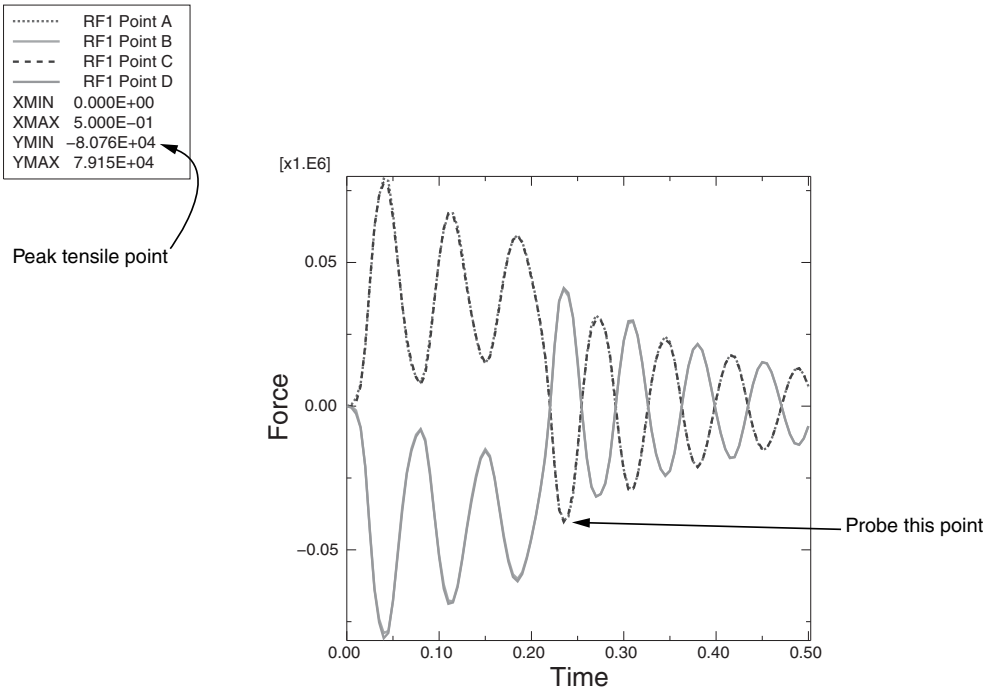


Figure 7–9 History of the reaction forces at the attached nodes.

To query the *X–Y* plot:

1. From the main menu bar, select **Tools→Query**.
   The **Query** dialog box appears.
2. Click **Probe values** in the **Visualization Module Queries** field.
   The **Probe Values** dialog box appears.
3. Select the point indicated in Figure 7–9.
   The *Y*-coordinate of this point is –40.30 kN, which corresponds to the value of the reaction force in the 1-direction.

7.6 Effect of the number of modes

For this simulation 30 modes were used to represent the dynamic behavior of the structure. The total modal effective mass for all of these modes was well over 90% of the mass of the structure that can move in the y- and z-directions, indicating that the dynamic representation is adequate.

Figure 7–10 shows the displacement in the direction of degree of freedom 2 at node 104 versus time and illustrates the effect of using fewer modes on the quality of the results. If you look at the table of effective mass, you will see that the first significant mode in the 2-direction is mode 3, which accounts for the lack of response when only two modes are used. The displacement of this node in the direction of degree of freedom 2 for the analyses using five modes and 30 modes is similar after 0.2 seconds; however, the early response differs, suggesting that there are significant modes in the range of 5–30 relating to the early response. When five modes are used, the total modal effective mass in the 2-direction is only 57% of the moveable mass.

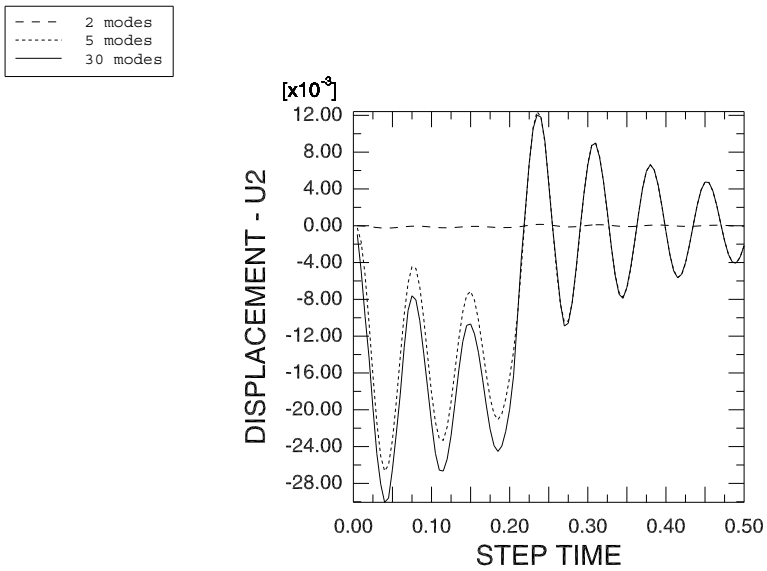


Figure 7–10 Effect of different numbers of modes on the results.

7.7 Effect of damping

In this simulation we used 5% of critical damping in all modes. This value was chosen from experience, based on the fact that the bolted connections between the trusses and the cross bracing might absorb

significant energy as a result of local frictional effects. In cases such as this, where accurate data are not available, it is important to investigate the effect of the choices that you make.

Figure 7–11 compares the history of the reaction force at one of the top connections (point C) when 1%, 5%, and 10% of critical damping are used. As expected, the oscillations at lower damping levels do not diminish as quickly as those at higher damping levels, and the peak force is higher in the models with lower damping. With damping ratios even as low as 1%, the peak pull-out force is 85 kN, which is still less than the strength of the connection (100 kN). Therefore, the cargo crane should retain its integrity under this drop load.

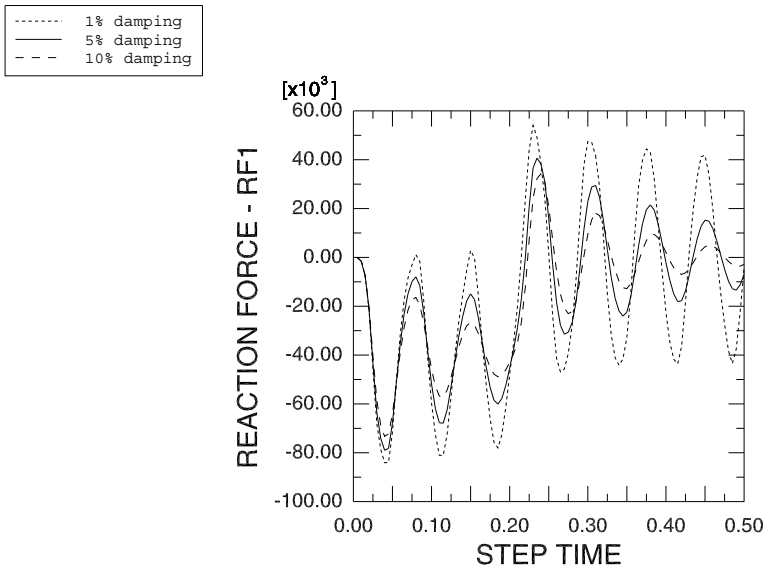


Figure 7–11 Effect of damping ratio on the pull-out force.

7.8 Comparison with direct time integration

Since this is a transient dynamic analysis, it is natural to consider how the results compare with those obtained using direct integration of the equations of motion. Direct integration can be performed with either implicit (Abaqus/Standard) or explicit (Abaqus/Explicit) methods. Here we extend the analysis to use the explicit dynamics procedure.

A direct comparison with the results presented earlier is not possible since the B33 element type and direct modal damping are not available in Abaqus/Explicit. Thus, in the Abaqus/Explicit analysis the element type is changed to B31 and Rayleigh damping is used in place of direct modal damping.

Save a copy of **dynamics.inp** as **dynamics_xpl.inp**. All subsequent changes should be made to **dynamics_xpl.inp**.

To modify the model:

1. Change the element type to B31 for all elements in the model. You can perform this change by modifying the TYPE parameter on the *ELEMENT option:

```
*ELEMENT, TYPE=B31
```

2. Add mass proportional damping to the bracing section properties. To do this, add the following *DAMPING option to the end of the material data option block for the bracing section:

```
*DAMPING, ALPHA=15.
```

This statement specifies a value of 15 for alpha damping and 0 for the remaining damping quantities. These values produce a reasonable trade-off in the values of critical damping at low and high frequencies of the structure. For the three lowest natural frequencies, the effective value of ξ is greater than 0.05, but as was shown in Figure 7–10, the first two modes do not contribute significantly to the response. For the remaining modes, the value of ξ is less than 0.05. The variation of ξ as a function of natural frequency is shown in Figure 7–12.

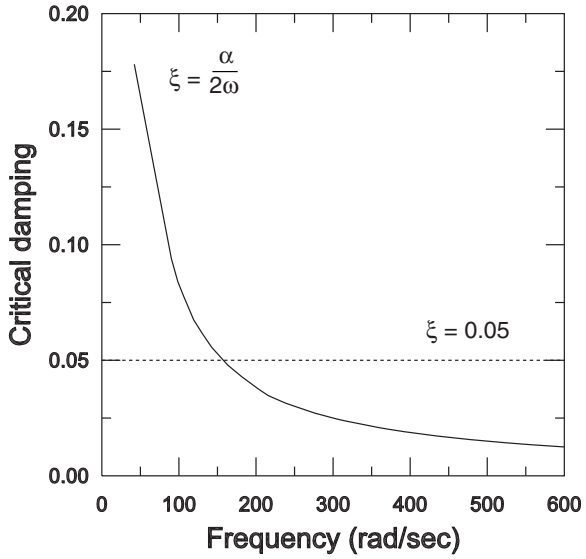


Figure 7–12 Effect of damping on the results.

3. Repeat the previous step for the main member section properties.

4. Delete both analysis steps.
5. Create an single explicit dynamics step, and specify a time period of **0.5** s. In addition, edit the step to use linear geometry by setting NLGEOM=NO on the *STEP option (this will result in a linear analysis). For your simulation the option block defining the explicit dynamics step should look similar to the following:

```
*STEP, NLGEOM=NO
Direct integration transient dynamic analysis
*DYNAMIC, EXPLICIT
, 0.5
*BULK VISCOSITY
0.06, 1.2
```

6. Redefine the tip load.

   The *CLOAD option block for this simulation is:

```
*CLOAD, AMPLITUDE=BOUNCE
104, 2, -1.0E4
```

7. Redefine node set **TIP** to include only node 104.

   Create a default field output request and two history output requests. In the first, request displacement history for the set **TIP**; in the second, request reaction force history for the set **ATTACH**.

   For your simulation, the option block defining the output requests should look similar to the following:

```
*NSET, NSET=TIP
104,
*OUTPUT, FIELD, VARIABLE=PRESELECT
*OUTPUT, HISTORY, VARIABLE=PRESELECT
*NODE OUTPUT, NSET=TIP
U,
*NODE OUTPUT, NSET=ATTACH
RF,
```

   Terminate the step with

```
*END STEP
```

8. Save the input file as **dynamics_xpl.inp** and submit for analysis:

```
abaqus job=dynamics_xpl.inp
```

When the job completes, navigate to the directory containing the output database file **dynamics_xpl.odb** and type the command

```
abaqus viewer odb=dynamics_xpl
```

at the operating system prompt to examine the results in Abaqus/Viewer. In particular, compare the tip displacement history obtained earlier from Abaqus/Standard with that obtained from Abaqus/Explicit. As shown in Figure 7–13, there are small differences in the response. These differences are due to the different element and damping types used for the modal dynamic analysis. In fact, if the Abaqus/Standard analysis is modified to use B31 elements and mass proportional damping, the results produced by the two analysis products are nearly indistinguishable (see Figure 7–13), which confirms the accuracy of the modal dynamic procedure.

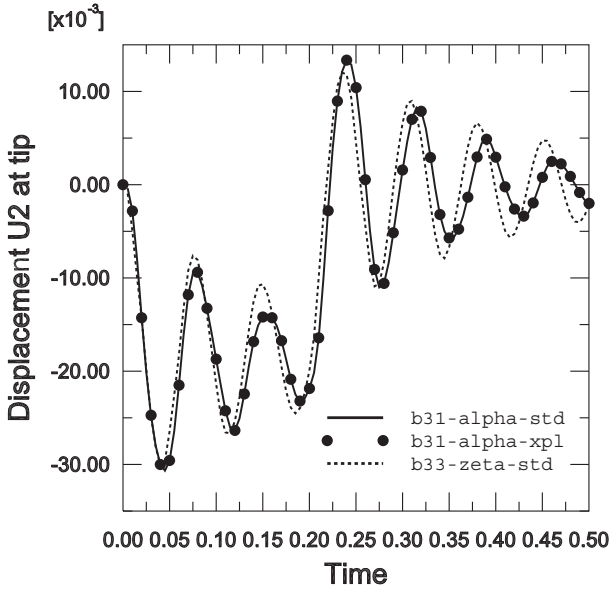


Figure 7–13 Comparison of tip displacements obtained from Abaqus/Standard and Abaqus/Explicit.

7.9 Other dynamic procedures

We now briefly review the other dynamic procedures available in Abaqus—namely linear modal dynamics and nonlinear dynamics.

7.9.1 Linear modal dynamics

There are several other linear, dynamic procedures in Abaqus/Standard that employ the modal superposition technique. Unlike *MODAL DYNAMIC, which calculates the response in the time domain, these procedures provide results in the frequency domain, which can give additional insight into the behavior of the structure.

A complete description of these procedures is given in "Dynamic stress/displacement analysis," Section 6.3 of the Abaqus Analysis User's Manual.

Steady-state dynamics

The *STEADY STATE DYNAMICS option calculates the amplitude and phase of the structure's response caused by harmonic excitation over a user-specified range of frequencies. Typical examples include the following:

- The response of car engine mounts over a range of engine operating speeds.
- Rotating machinery in buildings.
- Components on aircraft engines.

Response spectrum

The *RESPONSE SPECTRUM option provides an estimate of the peak response (displacement, stress, etc.) when a structure is subjected to dynamic motion of its fixed points. The motion of the fixed points is known as "base motion"; an example is a seismic event causing ground motion. Typically the method is used when an estimate of the peak response is required for design purposes.

Random response

The *RANDOM RESPONSE option predicts the response of a system subjected to random continuous excitation. The excitation is expressed in a statistical sense using a power spectral density function. Examples of random response analysis include the following:

- The response of an airplane to turbulence.
- The response of a structure to noise, such as that emitted by a jet engine.

7.9.2 Nonlinear dynamics

As mentioned earlier, the *MODAL DYNAMIC procedure is suitable only for linear problems. When nonlinear dynamic response is of interest, the equations of motion must be integrated directly. The direct-integration of the equations of motion is performed in Abaqus/Standard using an implicit dynamics procedure (*DYNAMIC). When this procedure is used, the mass, damping, and stiffness matrices are assembled and the equation of dynamic equilibrium is solved at each point in time. Since these operations are computationally intensive, direct-integration dynamics is more expensive than the modal methods.

Since the nonlinear dynamic procedure in Abaqus/Standard uses implicit time integration, it is suitable for nonlinear structural dynamics problems, for example, in which a sudden event initiates the dynamic response, such as an impact, or when the structural response involves large amounts of energy being dissipated by plasticity or viscous damping. In such studies the high frequency response, which is important initially, is damped out rapidly by the dissipative mechanisms in the model.

An alternative for nonlinear dynamic analyses is the explicit dynamics procedure available in Abaqus/Explicit. As discussed in Chapter 2, "Abaqus Basics," the explicit algorithm propagates the solution as a stress wave through the model, one element at a time. Thus, it is most suitable for applications in which stress wave effects are important and in which the event time being simulated is short (typically less than one second).

Another advantage associated with the explicit algorithm is that it can model discontinuous nonlinearities such as contact and failure more easily than Abaqus/Standard. Large, highly discontinuous problems are often more easily modeled with Abaqus/Explicit, even if the response is quasi-static. Explicit dynamic analyses are discussed further in Chapter 9, "Nonlinear Explicit Dynamics."

7.10 Related Abaqus examples

- "Linear analysis of the Indian Point reactor feedwater line," Section 2.2.2 of the Abaqus Example Problems Manual
- "Explosively loaded cylindrical panel," Section 1.3.3 of the Abaqus Benchmarks Manual
- "Eigenvalue analysis of a cantilever plate," Section 1.4.6 of the Abaqus Benchmarks Manual

7.11 Suggested reading

- Clough, R. W. and J. Penzien, *Dynamics of Structures*, McGraw-Hill, 1975.
- NAFEMS Ltd., *A Finite Element Dynamics Primer*, 1993.
- Spence, P. W. and C. J. Kenchington, *The Role of Damping in Finite Element Analysis*, Report R0021, NAFEMS Ltd., 1993.

7.12 Summary

- Dynamic analyses include the effect of the structure's inertia.
- The *FREQUENCY procedure extracts the natural frequencies and mode shapes of the structure.
- The mode shapes can then be used to determine the dynamic response of linear systems by modal superposition. This technique is efficient, but it cannot be used for nonlinear problems.

- Linear dynamic procedures are available in Abaqus/Standard to calculate the transient response to transient loading, the steady-state response to harmonic loading, the peak response to base motion, and the response to random loading.
- You should extract enough modes to obtain an accurate representation of the dynamic behavior of the structure. The total modal effective mass in the direction in which motion will occur should be at least 90% of the mass that can move to produce accurate results.
- You can define direct modal damping, Rayleigh damping, and composite modal damping in Abaqus/Standard. However, since the natural frequencies and mode shapes are based on the undamped structure, the structure being analyzed should be only lightly damped.
- Modal techniques are not suitable for nonlinear dynamic simulations. Direct time integration (*DYNAMIC) methods must be used in these situations.
- The *AMPLITUDE option allows any time variation of loads or prescribed boundary conditions to be defined.
- Mode shapes and transient results can be animated in Abaqus/Viewer. This provides a useful way of understanding the response of dynamic and nonlinear static analyses.

8. Nonlinearity

This chapter discusses nonlinear structural analysis in Abaqus. The differences between linear and nonlinear analyses are summarized below.

Linear analysis

All the analyses discussed so far have been linear: there is a linear relationship between the applied loads and the response of the system. For example, if a linear spring extends statically by 1 m under a load of 10 N, it will extend by 2 m when a load of 20 N is applied. This means that in a linear Abaqus/Standard analysis the flexibility of the structure need only be calculated once (by assembling the stiffness matrix and inverting it). The linear response of the structure to other load cases can be found by multiplying the new vector of loads by the inverted stiffness matrix. Furthermore, the structure's response to various load cases can be scaled by constants and/or superimposed on one another to determine its response to a completely new load case, provided that the new load case is the sum (or multiple) of previous ones. This principle of superposition of load cases assumes that the same boundary conditions are used for all the load cases.

Abaqus/Standard uses the principle of superposition of load cases in linear dynamics simulations, which are discussed in Chapter 7, "Linear Dynamics."

Nonlinear analysis

A nonlinear structural problem is one in which the structure's stiffness changes as it deforms. All physical structures exhibit nonlinear behavior. Linear analysis is a convenient approximation that is often adequate for design purposes. It is obviously inadequate for many structural simulations including manufacturing processes, such as forging or stamping; crash analyses; and analyses of rubber components, such as tires or engine mounts. A simple example is a spring with a nonlinear stiffening response (see Figure 8–1).

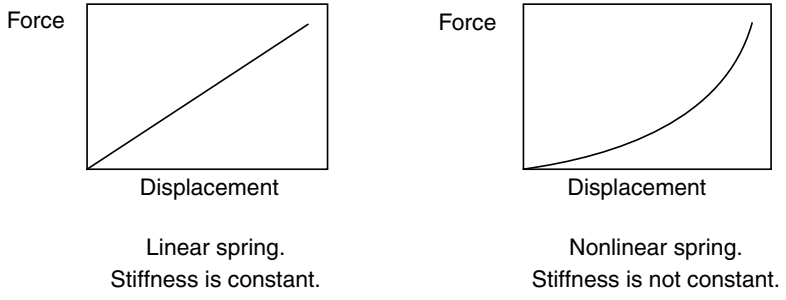


Figure 8–1 Linear and nonlinear spring characteristics.

Since the stiffness is now dependent on the displacement, the initial flexibility can no longer be multiplied by the applied load to calculate the spring's displacement for any load. In a nonlinear implicit analysis the stiffness matrix of the structure has to be assembled and inverted many times during the course of the analysis, making it much more expensive to solve than a linear implicit analysis. In an explicit analysis the increased cost of a nonlinear analysis is due to reductions in the stable time increment. The stable time increment is discussed further in Chapter 9, "Nonlinear Explicit Dynamics."

Since the response of a nonlinear system is not a linear function of the magnitude of the applied load, it is not possible to create solutions for different load cases by superposition. Each load case must be defined and solved as a separate analysis.

8.1 Sources of nonlinearity

There are three sources of nonlinearity in structural mechanics simulations:

- Material nonlinearity.
- Boundary nonlinearity.
- Geometric nonlinearity.

8.1.1 Material nonlinearity

This type of nonlinearity is probably the one that you are most familiar with and is covered in more depth in Chapter 10, "Materials." Most metals have a fairly linear stress/strain relationship at low strain values; but at higher strains the material yields, at which point the response becomes nonlinear and irreversible (see Figure 8–2).

Rubber materials can be approximated by a nonlinear, reversible (elastic) response (see Figure 8–3).

Material nonlinearity may be related to factors other than strain. Strain-rate-dependent material data and material failure are both forms of material nonlinearity. Material properties can also be a function of temperature and other predefined fields.

8.1.2 Boundary nonlinearity

Boundary nonlinearity occurs if the boundary conditions change during the analysis. Consider the cantilever beam, shown in Figure 8–4, that deflects under an applied load until it hits a "stop."

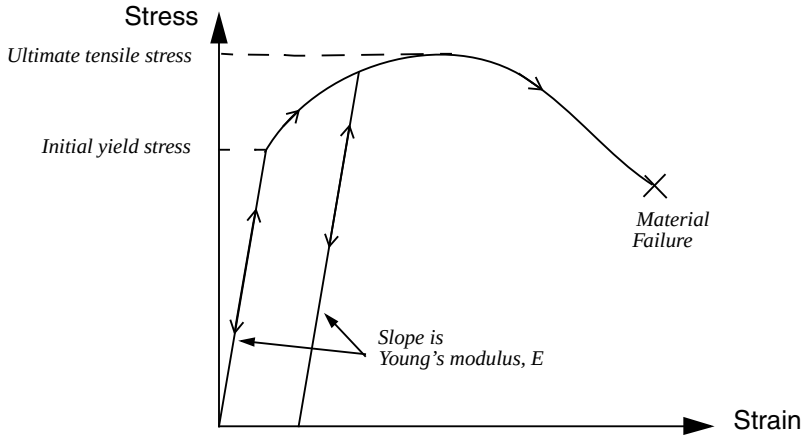


Figure 8–2 Stress-strain curve for an elastic-plastic material under uniaxial tension.

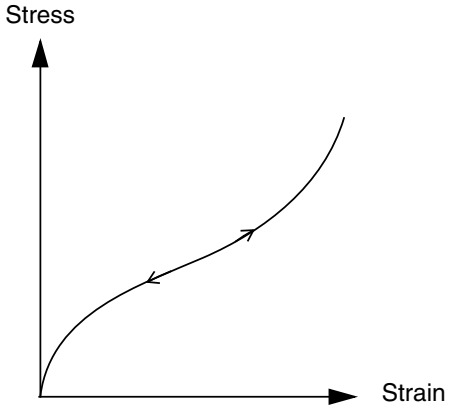


Figure 8–3 Stress-strain curve for a rubber-type material.

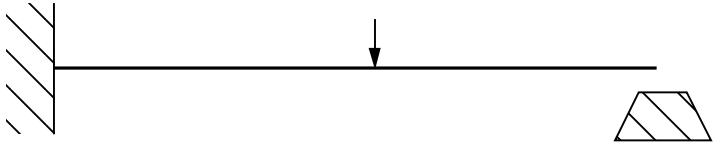


Figure 8–4 Cantilever beam hitting a stop.

The vertical deflection of the tip is linearly related to the load (if the deflection is small) until it contacts the stop. There is then a sudden change in the boundary condition at the tip of the beam, preventing any further vertical deflection, and so the response of the beam is no longer linear. Boundary nonlinearities are extremely discontinuous: when contact occurs during a simulation, there is a large and instantaneous change in the response of the structure.

Another example of boundary nonlinearity is blowing a sheet of material into a mold. The sheet expands relatively easily under the applied pressure until it begins to contact the mold. From then on the pressure must be increased to continue forming the sheet because of the change in boundary conditions.

Boundary nonlinearity is covered in Chapter 12, "Contact."

8.1.3 Geometric nonlinearity

The third source of nonlinearity is related to changes in the geometry of the structure during the analysis. Geometric nonlinearity occurs whenever the magnitude of the displacements affects the response of the structure. This may be caused by:

- Large deflections or rotations.
- "Snap through."
- Initial stresses or load stiffening.

For example, consider a cantilever beam loaded vertically at the tip (see Figure 8–5).

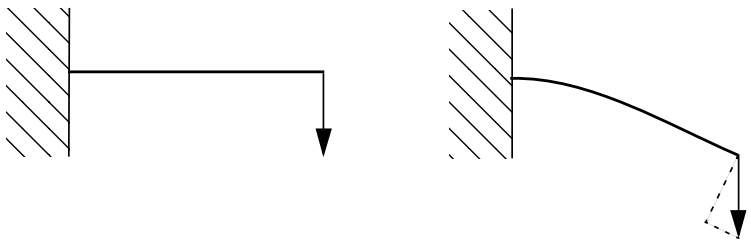


Figure 8–5 Large deflection of a cantilever beam.

If the tip deflection is small, the analysis can be considered as being approximately linear. However, if the tip deflections are large, the shape of the structure and, hence, its stiffness changes. In addition, if the load does not remain perpendicular to the beam, the action of the load on the structure changes significantly. As the cantilever beam deflects, the load can be resolved into a component perpendicular to the beam and a component acting along the length of the beam. Both of these effects contribute to the nonlinear response of the cantilever beam (i.e., the changing of the beam's stiffness as the load it carries increases).

One would expect large deflections and rotations to have a significant effect on the way that structures carry loads. However, displacements do not necessarily have to be large relative to the

dimensions of the structure for geometric nonlinearity to be important. Consider the "snap through" under applied pressure of a large panel with a shallow curve, as shown in Figure 8–6.

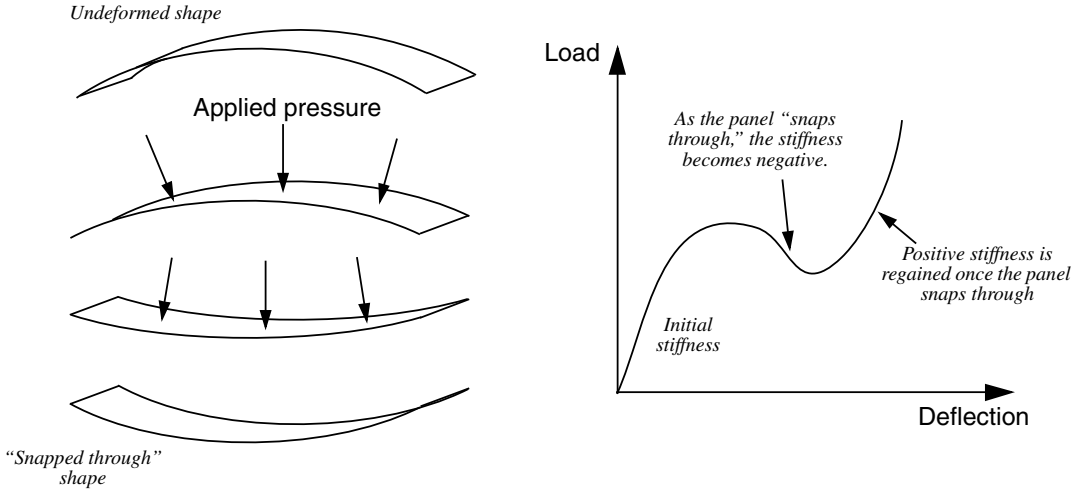


Figure 8–6 Snap-through of a large shallow panel.

In this example there is a dramatic change in the stiffness of the panel as it deforms. As the panel "snaps through," the stiffness becomes negative. Thus, although the magnitude of the displacements, relative to the panel's dimensions, is quite small, there is significant geometric nonlinearity in the simulation, which must be taken into consideration.

An important difference between the analysis products should be noted here: by default, Abaqus/Standard assumes small deformations, while Abaqus/Explicit assumes large deformations.

8.2 The solution of nonlinear problems

The nonlinear load-displacement curve for a structure is shown in Figure 8–7. The objective of the analysis is to determine this response. Consider the external forces, P, and the internal (nodal) forces, I, acting on a body (see Figure 8–8(a) and Figure 8–8(b), respectively). The internal loads acting on a node are caused by the stresses in the elements that contain that node.

For the body to be in static equilibrium, the net force acting at every node must be zero. Therefore, the basic statement of static equilibrium is that the internal forces, I, and the external forces, P, must balance each other:

$$P - I = 0.$$

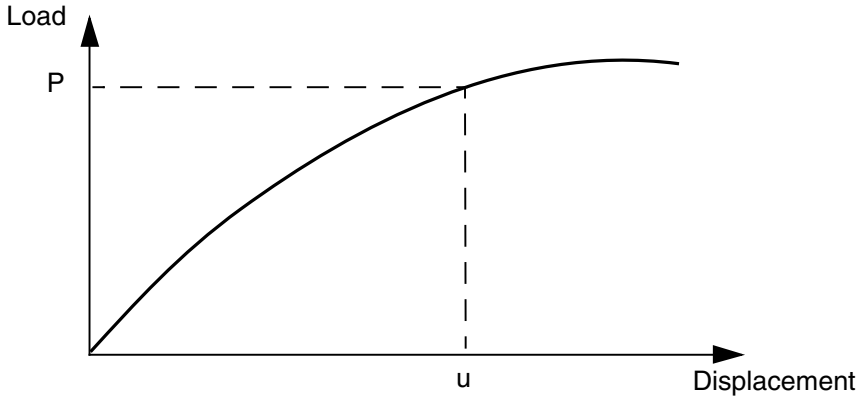
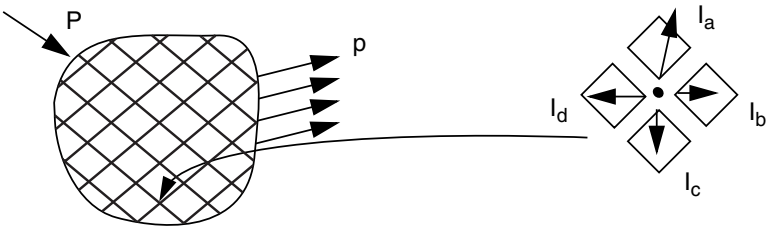


Figure 8–7 Nonlinear load-displacement curve.



(a) External loads in a simulation. (b) Internal forces acting at a node.

Figure 8–8 Internal and external loads on a body.

Abaqus/Standard uses the Newton-Raphson method to obtain solutions for nonlinear problems. In a nonlinear analysis the solution usually cannot be calculated by solving a single system of equations, as would be done in a linear problem. Instead, the solution is found by applying the specified loads gradually and incrementally working toward the final solution. Therefore, Abaqus/Standard breaks the simulation into a number of *load increments* and finds the approximate equilibrium configuration at the end of each load increment. It often takes Abaqus/Standard several iterations to determine an acceptable solution to a given load increment. The sum of all of the incremental responses is the approximate solution for the nonlinear analysis. Thus, Abaqus/Standard combines incremental and iterative procedures for solving nonlinear problems.

Abaqus/Explicit determines a solution to the dynamic equilibrium equation $P - I = M\ddot{u}$ without iterating by explicitly advancing the kinematic state from the end of the previous increment. Solving a problem explicitly does not require the formation of tangent stiffness matrices. The explicit central-difference operator satisfies the dynamic equilibrium equations at the beginning of the increment, t; the accelerations calculated at time t are used to advance the velocity solution to time $t + \Delta t/2$ and the displacement solution to time $t + \Delta t$. For linear and nonlinear problems alike, explicit methods require a small time increment size that depends solely on the highest natural frequency of the model and is independent of the type and duration of loading. Simulations typically require a large number of increments; however, due to the fact that a global set of equations is not solved in each increment, the cost per increment of an explicit method is much smaller than that of an implicit method. The small increments characteristic of an explicit dynamic method make Abaqus/Explicit well suited for nonlinear analysis.

8.2.1 Steps, increments, and iterations

This section introduces some new vocabulary for describing the various parts of an analysis. It is important that you clearly understand the difference between an analysis *step*, a load *increment*, and an *iteration*.

- The load history for a simulation consists of one or more steps. You define the steps, which generally consist of an analysis procedure option, loading options, and output request options. Different loads, boundary conditions, analysis procedure options, and output requests can be used in each step. For example:
  - Step 1: Hold a plate between rigid jaws.
  - Step 2: Add loads to deform the plate.
  - Step 3: Find the natural frequencies of the deformed plate.
- An increment is part of a step. In nonlinear analyses the total load applied in a step is broken into smaller increments so that the nonlinear solution path can be followed.

  In Abaqus/Standard you suggest the size of the first increment, and Abaqus/Standard automatically chooses the size of the subsequent increments. In Abaqus/Explicit the default time incrementation is fully automatic and does not require user intervention. Because the explicit method is conditionally stable, there is a stability limit for the time increment. The stable time increment is discussed in Chapter 9, "Nonlinear Explicit Dynamics."

  At the end of each increment the structure is in (approximate) equilibrium and results are available for writing to the output database, restart, data, or results files. The increments at which you select results to be written to the output database file are called *frames*.

  The issues associated with time incrementation in Abaqus/Standard and Abaqus/Explicit analyses are quite different, since time increments are generally much smaller in Abaqus/Explicit.
- An iteration is an attempt at finding an equilibrium solution in an increment when solving with an implicit method. If the model is not in equilibrium at the end of the iteration, Abaqus/Standard

tries another iteration. With every iteration the solution Abaqus/Standard obtains should be closer to equilibrium; sometimes Abaqus/Standard may need many iterations to obtain an equilibrium solution. When an equilibrium solution has been obtained, the increment is complete. Results can be requested only at the end of an increment.

Abaqus/Explicit does not need to iterate to obtain the solution in an increment.

8.2.2 Equilibrium iterations and convergence in Abaqus/Standard

The nonlinear response of a structure to a small load increment, ΔP, is shown in Figure 8–9. Abaqus/Standard uses the structure's initial stiffness, K_0, which is based on its configuration at u_0, and ΔP to calculate a *displacement correction*, c_a, for the structure. Using c_a, the structure's configuration is updated to u_a.

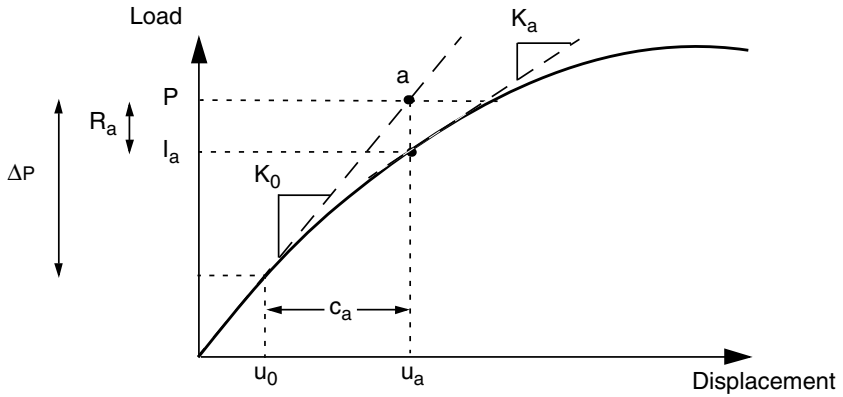


Figure 8–9 First iteration in an increment.

Convergence

Abaqus/Standard forms a new stiffness, K_a, for the structure, based on its updated configuration, u_a. Abaqus/Standard also calculates I_a, in this updated configuration. The difference between the total applied load, P, and I_a can now be calculated as:

$$R_a = P - I_a,$$

where R_a is the *force residual* for the iteration.

If R_a is zero at every degree of freedom in the model, point a in Figure 8–9 would lie on the load-deflection curve, and the structure would be in equilibrium. In a nonlinear problem it is almost impossible to have R_a equal zero, so Abaqus/Standard compares it to a tolerance value. If R_a is less than this force residual tolerance, Abaqus/Standard accepts the structure's updated configuration

as the equilibrium solution. By default, this tolerance value is set to 0.5% of an average force in the structure, averaged over time. Abaqus/Standard automatically calculates this spatially and time-averaged force throughout the simulation.

If R_a is less than the current tolerance value, $\boldsymbol{P}$ and I_a are in equilibrium, and u_a is a valid equilibrium configuration for the structure under the applied load. However, before Abaqus/Standard accepts the solution, it also checks that the displacement correction, c_a, is small relative to the total incremental displacement, $\Delta u_a = u_a - u_0$. If c_a is greater than 1% of the incremental displacement, Abaqus/Standard performs another iteration. Both convergence checks must be satisfied before a solution is said to have *converged* for that load increment. The exception to this rule is the case of a *linear* increment, which is defined as any increment in which the largest force residual is less than 10^{-8} times the time-averaged force. Any case that passes such a stringent comparison of the largest force residual with the time-averaged force is considered linear and does not require further iteration. The solution is accepted without any check on the size of the displacement correction.

If the solution from an iteration is not converged, Abaqus/Standard performs another iteration to try to bring the internal and external forces into balance. This second iteration uses the stiffness, K_a, calculated at the end of the previous iteration together with R_a to determine another displacement correction, c_b, that brings the system closer to equilibrium (point $\boldsymbol{b}$ in Figure 8–10).

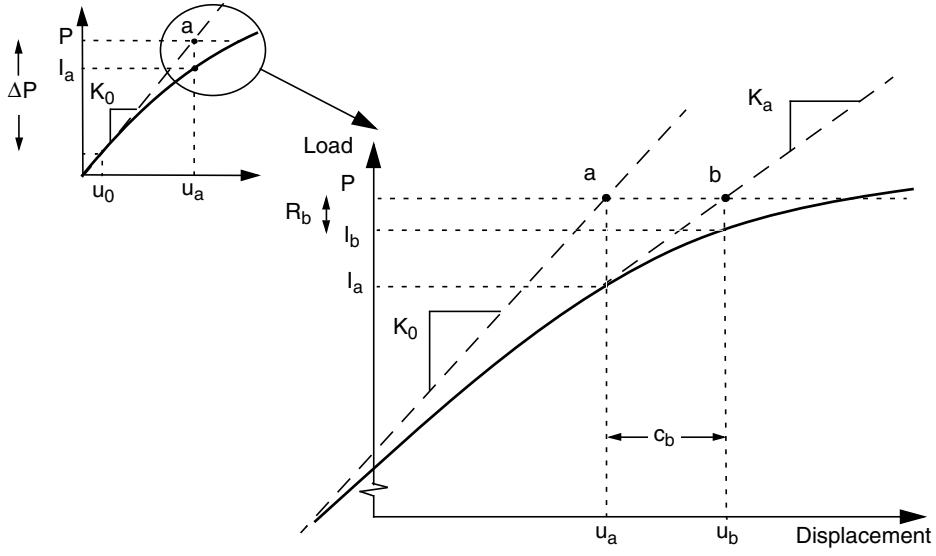


Figure 8–10 Second iteration.

Abaqus/Standard calculates a new force residual, R_b, using the internal forces from the structure's new configuration, u_b. Again, the largest force residual at any degree of freedom, R_b, is compared against the force residual tolerance, and the displacement correction for the second iteration, c_b, is compared to the increment of displacement, $\Delta u_b = u_b - u_0$. If necessary, Abaqus/Standard performs further iterations.

For each iteration in a nonlinear analysis Abaqus/Standard forms the model's stiffness matrix and solves a system of equations. This means that each iteration is equivalent, in computational cost, to conducting a complete linear analysis. It should now be clear that the computational expense of a nonlinear analysis in Abaqus/Standard can be many times greater than for a linear one.

It is possible with Abaqus/Standard to save results at each converged increment. Thus, the amount of output data available from a nonlinear simulation is many times that available from a linear analysis of the same geometry. Consider both of these factors and the types of nonlinear simulations that you want to perform when planning your computer resources.

8.2.3 Automatic incrementation control in Abaqus/Standard

Abaqus/Standard automatically adjusts the size of the load increments so that it solves nonlinear problems easily and efficiently. You only need to suggest the size of the first increment in each step of your simulation. Thereafter, Abaqus/Standard automatically adjusts the size of the increments. If you do not provide a suggested initial increment size, Abaqus/Standard will try to apply all of the loads defined in the step in the first increment. In highly nonlinear problems Abaqus/Standard will have to reduce the increment size repeatedly, resulting in wasted CPU time. Generally it is to your advantage to provide a reasonable initial increment size (see "Modifications to the input file—the history data," Section 8.4.1, for an example); only in very mildly nonlinear problems can all of the loads in a step be applied in a single increment.

The number of iterations needed to find a converged solution for a load increment will vary depending on the degree of nonlinearity in the system. By default, if the solution appears to diverge, Abaqus/Standard abandons the increment and starts again with the increment size set to 25% of its previous value. An attempt is then made at finding a converged solution with this smaller load increment. If the increment still fails to converge, Abaqus/Standard reduces the increment size again. By default, Abaqus/Standard allows a maximum of five cutbacks of increment size in an increment before stopping the analysis.

In Abaqus/Standard you can also add the INC parameter to specify the maximum number of increments allowed during the step. Abaqus/Standard terminates the analysis with an error message if it needs more increments than this limit to complete the step. The default number of increments for a step is 100; if significant nonlinearity is present in the simulation, the analysis may require many more increments. The INC parameter specifies an upper limit on the number of increments that Abaqus/Standard can use, rather than the number of increments it must use. For example, a step involving nonlinear geometry with a maximum of 150 increments would be specified as:

```
*STEP, NLGEOM=YES, INC=150
```

In a nonlinear analysis a step takes place over a finite period of "time," although this "time" has no physical meaning unless inertial effects or rate-dependent behavior are present. In Abaqus/Standard you specify the initial time increment, $\Delta T_{initial}$, and the total step time, T_{total} on the data line of the procedure option used in the step. For example,

```
*STATIC
0.1, 1.0
```

Total step time (T_{total})

Initial time increment ($\Delta T_{initial}$)

defines a static analysis that occurs over 1.0 units of time and has an initial increment of 0.1. The ratio of the initial time increment to the step time specifies the proportion of load applied in the first increment. The initial load increment is given by

$$\frac{\Delta T_{initial}}{T_{total}} \times \text{Load magnitude.}$$

The choice of initial time increment can be critical in certain nonlinear simulations in Abaqus/Standard, but for most analyses an initial increment size that is 5% to 10% of the total step time is usually sufficient. In static simulations the total step time is usually set to 1.0 for convenience, unless, for example, rate-dependent material effects or dashpots are included in the model. With a total step time of 1.0 the proportion of load applied is always equal to the current step time; i.e., 50% of the total load is applied when the step time is 0.5.

Although you must specify the initial increment size in Abaqus/Standard, Abaqus/Standard automatically controls the size of the subsequent increments. This automatic control of the increment size is suitable for the majority of nonlinear simulations performed with Abaqus/Standard, although further controls on the increment size are available. Abaqus/Standard will terminate an analysis if excessive cutbacks caused by convergence problems reduce the increment size below the minimum value. The default minimum allowable time increment, ΔT_{min}, is 10^{-5} times the total step time. By default, Abaqus/Standard has no upper limit on the increment size, ΔT_{max}, other than the total step time. Depending on your Abaqus/Standard simulation, you may want to specify different minimum and/or maximum allowable increment sizes. For example, if you know that your simulation may have trouble obtaining a solution if too large a load increment is applied, perhaps because the model may undergo plastic deformation, you may want to decrease ΔT_{max}.

If the increment converges in fewer than five iterations, this indicates that the solution is being found fairly easily. Therefore, Abaqus/Standard automatically increases the increment size by 50% if two consecutive increments require fewer than five iterations to obtain a converged solution.

Details of the automatic load incrementation scheme are given in the message file, as shown in more detail in "Results," Section 8.4.3.

8.3 Including nonlinearity in an Abaqus analysis

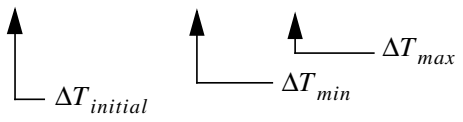
We now discuss how to account for nonlinearity in an Abaqus analysis. The main focus is on geometric nonlinearity.

8.3.1 Geometric nonlinearity

Incorporating the effects of geometric nonlinearity in an analysis requires only minor changes to an Abaqus/Standard model. You need to make sure the step definition considers geometrically nonlinear effects by setting the NLGEOM parameter equal to YES on the *STEP option. This is the default setting in Abaqus/Explicit. You also need to set time incrementation parameters as discussed in "Automatic incrementation control in Abaqus/Standard," Section 8.2.3.

The following input describes a static analysis in which the load is applied over 5 units of time and the initial time increment is 1 unit of time; the minimum and maximum time increments are set to 0.0001 and 1.5, respectively:

```
*STATIC
1.0, 5.0, 0.0001, 1.5
```



Abaqus will apply 20% (1.0/5.0) of the total load in the first increment, and it will terminate the analysis if it has problems converging and requires an increment smaller than 0.0001. If the time increment grows because the solution is converging easily, the maximum time increment Abaqus can use is 1.5.

Local directions

In a geometrically nonlinear analysis the local material directions may rotate with the deformation in each element. For shell, beam, and truss elements the local material directions always rotate with the deformation. For solid elements the local material directions rotate with the deformation only if the elements refer to an *ORIENTATION option; otherwise, the default local material directions remain constant throughout the analysis.

Local directions defined at nodes by using the *TRANSFORM option remain fixed throughout the analysis; they do not rotate with the deformation. See "Transformed coordinate systems," Section 2.1.5 of the Abaqus Analysis User's Manual, for further details.

Effect on subsequent steps

Once you include geometric nonlinearity in a step, it is considered in all subsequent steps. If nonlinear geometric effects are not requested in a subsequent step, Abaqus will issue a warning stating that they are being included in the step anyway.

Other geometrically nonlinear effects

The large deformations in a model are not the only important effects that are considered when geometric nonlinearity is activated. Abaqus/Standard also includes terms in the element stiffness calculations that are caused by the applied loads, the so-called load stiffness. These terms improve convergence behavior. In addition, the membrane loads in shells and the axial loads in cables and beams contribute much of the stiffness of these structures in response to transverse loads. By including geometric nonlinearity, the membrane stiffness in response to transverse loads is considered as well.

8.3.2 Material nonlinearity

The addition of material nonlinearity to an Abaqus model is discussed in Chapter 10, "Materials."

8.3.3 Boundary nonlinearity

The introduction of boundary nonlinearity is discussed in Chapter 12, "Contact."

8.4 Example: nonlinear skew plate

This example is a continuation of the linear skew plate simulation described in Chapter 5, "Using Shell Elements," and shown in Figure 8–11. You will now reanalyze the plate in Abaqus/Standard to include the effects of geometric nonlinearity. The results from this analysis will allow you to determine the importance of geometrically nonlinear effects and, therefore, the validity of the linear analysis.

You only need to modify the history data to convert this model from a linear simulation to a nonlinear simulation.

If you wish, you can follow the guidelines at the end of this example to extend the simulation to perform a dynamic analysis using Abaqus/Explicit.

The steps that follow assumes that you have access to the full input file for this example. This input file, **`skew_nl.inp`**, is available in "Nonlinear skew plate," Section A.6, in the online HTML version of this manual. Instructions on how to fetch and run the script are given in Appendix A, "Example Files." If you wish to create the entire model using Abaqus/CAE, please refer to "Example: nonlinear skew plate," Section 8.4 of Getting Started with Abaqus: Interactive Edition.

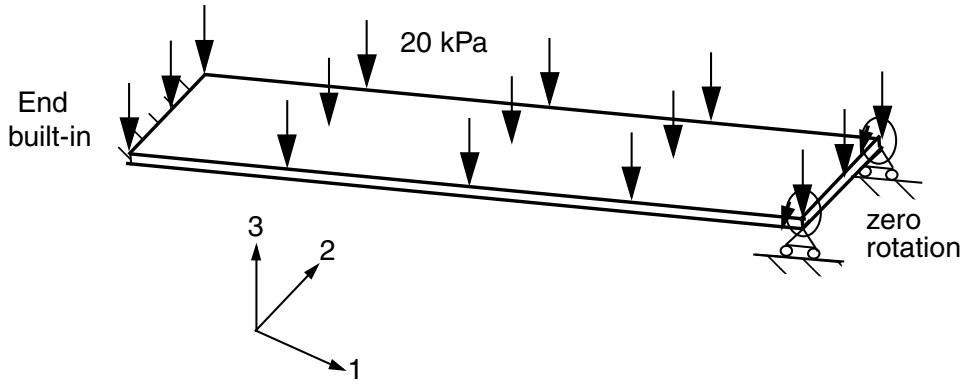


Figure 8–11 Skew plate.

8.4.1 Modifications to the input file—the history data

This example does not change any model data in the original skew plate example; only history data changes are included.

Applying NLGEOM to the step

Set the NLGEOM parameter equal to YES on the ∗STEP option, and remove the PERTURBATION parameter. This indicates that the analysis now includes nonlinear geometric effects. The default maximum number of increments is 100; Abaqus may use fewer increments than this upper limit, but it will stop the analysis if it needs more.

The modified ∗STEP option looks like:

```
*STEP, NLGEOM=YES
```

You may also wish to modify the description of the step to reflect that this is now a nonlinear analysis.

Defining the step time

This analysis requires a data line to the ∗STATIC option that specifies the size of the initial time increment, $\Delta T_{initial}$, for the analysis and the total step time for the simulation. A total step time of 1.0 is used, and $\Delta T_{initial}$ is specified such that Abaqus applies 10% of the load in the first increment. The completed ∗STATIC option block will, therefore, be

```
*STATIC
0.1, 1.0
```

Output control

In a linear analysis Abaqus solves the equilibrium equations once and calculates the results for this one solution. A nonlinear analysis can produce much more output because results can be requested at the end of each converged increment. If you do not select the output requests carefully, the output files become very large, potentially filling the disk space on your computer.

As noted earlier, output is available in four different files:

- the output database (**.odb**) file, which contains data in a neutral binary format necessary to postprocess the results with Abaqus/Viewer;
- the data (**.dat**) file, which contains printed tables of selected results;
- the restart (**.res**) file, which is used to continue the analysis; and
- the results (**.fil**) file, which is used with third-party postprocessors.

Only the options for the output database (**.odb**) and printed output (**.dat**) files are discussed here. If selected carefully, data can be saved frequently during the simulation without using excessive disk space.

Remove the existing output requests from your input file; and add the following output options, which ensure that only selected output is saved during the nonlinear analysis.

To reduce the size of the output database file, the FREQUENCY parameter is used on the *OUTPUT, FIELD option; in this simulation field output is written every second increment. Thus, the FREQUENCY parameter is set to 2 on the *OUTPUT option:

```
*OUTPUT, FIELD, FREQUENCY=2, VARIABLE=PRESELECT
```

The parameter VARIABLE=PRESELECT indicates that a preselected set of the most commonly used field variables for a given type of analysis will be written to the output database (**.odb**) file. If you are simply interested in the final results, set the FREQUENCY parameter equal to a large number. Results are always stored at the end of each step, regardless of the value of the FREQUENCY parameter; therefore, using a large value causes only the final results to be saved.

Request that the displacements of the nodes at the midspan be saved to the output database file. These results will be used later to demonstrate the *X–Y* plotting capability in Abaqus/Viewer. Use the default FREQUENCY value (FREQUENCY=1) for this history output request for the output database file. Here the *NODE OUTPUT option must appear immediately after the history output request. Remember to use the NSET or ELSET parameters when limiting the output being requested to a subset of the model; otherwise, the default subset, which is the entire model, will be used.

Finally, request that reaction forces (RF) be printed for all the nodes in the model and that the displacements (U) be printed for the nodes at the midspan (node set **MIDSPAN**). You will need two *NODE PRINT options because you are requesting results for two different subsets of the model. Again, use the FREQUENCY parameter to reduce the amount of output; print the data every second increment.

The new list of output request option blocks looks like:

```
*OUTPUT, FIELD, FREQUENCY=2, VARIABLE=PRESELECT
*OUTPUT, HISTORY, FREQUENCY=1
*NODE OUTPUT, NSET=MIDSPAN
U,
*NODE PRINT, NSET=MIDSPAN, FREQUENCY=2
U,
*NODE PRINT, SUMMARY=NO, TOTALS=YES, FREQUENCY=2
RF,
```

Finally, the step definition is completed using the *END STEP option.

```
*END STEP
```

8.4.2 Running the analysis

Store the modified input in a file called **skew_nl.inp** (an example input file is listed in "Nonlinear skew plate," Section A.6). Run the analysis using the following command:

```
abaqus job=skew_nl interactive
```

8.4.3 Results

During a nonlinear analysis, two additional output files become very important. They are the status file (**skew_nl.sta**) and the message file (**skew_nl.msg**). As the analysis progresses, Abaqus will write data to both of these files. You can view the data even as Abaqus continues your analysis. You will need to learn how to use the data in these files to assess Abaqus's progress on your simulations. There may be situations where you decide, based on information in these files, to terminate your analysis early. A more likely scenario is that you may need to use these files to learn what caused Abaqus to terminate your analysis prematurely; i.e., what caused the convergence problems.

Status file

The status file is particularly useful for monitoring the progress of a nonlinear simulation while the job is running. The output below shows the status file for this nonlinear skewed plate example.

```
SUMMARY OF JOB INFORMATION:
STEP   INC ATT SEVERE EQUIL TOTAL   TOTAL       STEP        INC OF        DOF     IF
               DISCON ITERS ITERS   TIME/     TIME/LPF    TIME/LPF    MONITOR RIKS
               ITERS                FREQ
  1     1   1     0     4     4   0.100       0.100       0.1000
  1     2   1     0     2     2   0.200       0.200       0.1000
  1     3   1     0     2     2   0.350       0.350       0.1500
  1     4   1     0     2     2   0.575       0.575       0.2250
  1     5   1     0     3     3   0.913       0.913       0.3375
  1     6   1     0     2     2   1.00        1.00        0.08750
```

The status file contains a separate line for every converged increment in the simulation. The first column shows the step number—in this case there is only one step. The second column gives the

increment number. The sixth column shows the number of iterations Abaqus needed to obtain a converged solution in each increment; for example, Abaqus needed 4 iterations in increment 1. The eighth column shows the total step time completed, and the ninth column shows the increment size (ΔT).

This example shows how Abaqus automatically controls the increment size and, therefore, the proportion of load applied in each increment. In this analysis Abaqus applied 10% of the total load in the first increment: you specified $\Delta T_{initial}$ to be 0.1 and the step time to be 1.0. Abaqus needed four iterations to converge to a solution in the first increment. Abaqus only needed two iterations in the second increment, so it automatically increased the size of the next increment by 50% to ΔT = 0.15. Abaqus also increased ΔT in both the fourth and fifth increments. It adjusted the final increment size to be just enough to complete the analysis; in this case the final increment size was 0.0875.

Message file

The message file contains more detailed information about the progress of the analysis than the status file. Abaqus lists all of the tolerances and parameters to control the analysis at the start of each step in the message file, as shown below. This is done for each step because these controls can be modified from step to step. The default values of these controls are appropriate for most analyses, so normally it is not necessary for you to modify them. The modification of control tolerances and parameters is beyond the scope of this guide (it is discussed in "Commonly used control parameters," Section 7.2.2 of the Abaqus Analysis User's Manual).

```
                    S T E P       1     S T A T I C   A N A L Y S I S


         Uniform pressure (20.0 kPa) load

     AUTOMATIC TIME CONTROL WITH -
          A SUGGESTED INITIAL TIME INCREMENT OF                0.100
          AND A TOTAL TIME PERIOD OF                           1.00
          THE MINIMUM TIME INCREMENT ALLOWED IS                1.000E-05
          THE MAXIMUM TIME INCREMENT ALLOWED IS                1.00

     LINEAR EQUATION SOLVER TYPE         DIRECT SPARSE

 CONVERGENCE TOLERANCE PARAMETERS FOR FORCE
     CRITERION FOR RESIDUAL FORCE     FOR A NONLINEAR PROBLEM          5.000E-03
     CRITERION FOR DISP.    CORRECTION IN A NONLINEAR PROBLEM          1.000E-02
     INITIAL VALUE OF TIME AVERAGE FORCE                               1.000E-02
     AVERAGE FORCE     IS TIME AVERAGE FORCE
     ALTERNATE CRIT. FOR RESIDUAL FORCE     FOR A NONLINEAR PROBLEM    2.000E-02
     CRITERION FOR ZERO FORCE     RELATIVE TO TIME AVRG. FORCE         1.000E-05
     CRITERION FOR RESIDUAL FORCE     WHEN THERE IS ZERO FLUX          1.000E-05
     CRITERION FOR DISP.    CORRECTION WHEN THERE IS ZERO FLUX          1.000E-03
     CRITERION FOR RESIDUAL FORCE     FOR A LINEAR INCREMENT           1.000E-08
     FIELD CONVERSION RATIO                                             1.00
     CRITERION FOR ZERO FORCE     REL. TO TIME AVRG. MAX. FORCE        1.000E-05
     CRITERION FOR ZERO DISP.    RELATIVE TO CHARACTERISTIC LENGTH     1.000E-08

 CONVERGENCE TOLERANCE PARAMETERS FOR MOMENT
     CRITERION FOR RESIDUAL MOMENT    FOR A NONLINEAR PROBLEM          5.000E-03
     CRITERION FOR ROTATION CORRECTION IN A NONLINEAR PROBLEM          1.000E-02
     INITIAL VALUE OF TIME AVERAGE MOMENT                              1.000E-02
     AVERAGE MOMENT    IS TIME AVERAGE MOMENT
     ALTERNATE CRIT. FOR RESIDUAL MOMENT    FOR A NONLINEAR PROBLEM    2.000E-02
     CRITERION FOR ZERO MOMENT    RELATIVE TO TIME AVRG. MOMENT        1.000E-05
```

```
    CRITERION FOR RESIDUAL MOMENT    WHEN THERE IS ZERO FLUX          1.000E-05
    CRITERION FOR ROTATION CORRECTION WHEN THERE IS ZERO FLUX         1.000E-03
    CRITERION FOR RESIDUAL MOMENT    FOR A LINEAR INCREMENT           1.000E-08
    FIELD CONVERSION RATIO                                             1.00
    CRITERION FOR ZERO MOMENT    REL. TO TIME AVRG. MAX. MOMENT       1.000E-05

    VOLUMETRIC STRAIN COMPATIBILITY TOLERANCE FOR HYBRID SOLIDS       1.000E-05
    AXIAL STRAIN COMPATIBILITY TOLERANCE FOR HYBRID BEAMS             1.000E-05
    TRANS. SHEAR STRAIN COMPATIBILITY TOLERANCE FOR HYBRID BEAMS      1.000E-05
    SOFT CONTACT CONSTRAINT COMPATIBILITY TOLERANCE FOR P>P0          5.000E-03
    SOFT CONTACT CONSTRAINT COMPATIBILITY TOLERANCE FOR P=0.0         0.100
    DISPLACEMENT COMPATIBILITY TOLERANCE FOR DCOUP ELEMENTS           1.000E-05
    ROTATION COMPATIBILITY TOLERANCE FOR DCOUP ELEMENTS               1.000E-05

EQUILIBRIUM WILL BE CHECKED FOR SEVERE DISCONTINUITY ITERATIONS

TIME INCREMENTATION CONTROL PARAMETERS:
    FIRST EQUILIBRIUM ITERATION FOR CONSECUTIVE DIVERGENCE CHECK              4
    EQUILIBRIUM ITERATION AT WHICH LOG. CONVERGENCE RATE CHECK BEGINS         8
    EQUILIBRIUM ITERATION AFTER WHICH ALTERNATE RESIDUAL IS USED              9
    MAXIMUM EQUILIBRIUM ITERATIONS ALLOWED                                   16
    EQUILIBRIUM ITERATION COUNT FOR CUT-BACK IN NEXT INCREMENT               10
    MAXIMUM EQUILIB. ITERS IN TWO INCREMENTS FOR TIME INCREMENT INCREASE      4
    MAXIMUM ITERATIONS FOR SEVERE DISCONTINUITIES                            50
    MAXIMUM CUT-BACKS ALLOWED IN AN INCREMENT                                 5
    MAXIMUM DISCON. ITERS IN TWO INCREMENTS FOR TIME INCREMENT INCREASE       50
    MAXIMUM CONTACT AUGMENTATIONS FOR *SURFACE BEHAVIOR,AUGMENTED LAGRANGE    6
    CUT-BACK FACTOR AFTER DIVERGENCE                                     0.2500
    CUT-BACK FACTOR FOR TOO SLOW CONVERGENCE                             0.5000
    CUT-BACK FACTOR AFTER TOO MANY EQUILIBRIUM ITERATIONS                0.7500
    CUT-BACK FACTOR AFTER TOO MANY SEVERE DISCONTINUITY ITERATIONS       0.2500
    CUT-BACK FACTOR AFTER PROBLEMS IN ELEMENT ASSEMBLY                   0.2500
    INCREASE FACTOR AFTER TWO INCREMENTS THAT CONVERGE QUICKLY             1.500
    MAX. TIME INCREMENT INCREASE FACTOR ALLOWED                            1.500
    MAX. TIME INCREMENT INCREASE FACTOR ALLOWED (DYNAMICS)                 1.250
    MAX. TIME INCREMENT INCREASE FACTOR ALLOWED (DIFFUSION)                2.000
    MINIMUM TIME INCREMENT RATIO FOR EXTRAPOLATION TO OCCUR              0.1000
    MAX. RATIO OF TIME INCREMENT TO STABILITY LIMIT                        1.000
    FRACTION OF STABILITY LIMIT FOR NEW TIME INCREMENT                   0.9500
    TIME INCREMENT INCREASE FACTOR BEFORE A TIME POINT                     1.000
    GLOBAL STABILIZATION CONTROL IS NOT USED

         PRINT OF INCREMENT NUMBER, TIME, ETC., EVERY    1  INCREMENTS
```

Abaqus lists a summary of each iteration in the message file after the lists of tolerances and controls. It prints the values of the largest residual force, r^{α}_{max}; largest increment of displacement, Δu_{α}; the largest correction to displacement, c_{α}; and the time averaged force, $\tilde{q}^{\alpha}$. It also prints the nodes and degrees of freedom (DOF) at which r^{α}_{max}, Δu_{α}, and c_{α} occur. A similar summary is printed for rotational degrees of freedom.

```
 INCREMENT     1 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  0.100

             CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1


AVERAGE FORCE                        12.2       TIME AVG. FORCE        12.2
LARGEST RESIDUAL FORCE              -749.       AT NODE       1051   DOF  1
LARGEST INCREMENT OF DISP.       -5.576E-03     AT NODE        559   DOF  3
LARGEST CORRECTION TO DISP.      -5.576E-03     AT NODE        559   DOF  3
         FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

AVERAGE MOMENT                       1.12       TIME AVG. MOMENT       1.12
LARGEST RESIDUAL MOMENT          -3.273E-03     AT NODE       1104   DOF  5
LARGEST INCREMENT OF ROTATION     1.598E-02     AT NODE        159   DOF  5
LARGEST CORRECTION TO ROTATION    1.598E-02     AT NODE        159   DOF  5
         ROTATION CORRECTION TOO LARGE COMPARED TO ROTATION INCREMENT
```

In this example the initial time increment is 0.1, as specified in the input file. The average force for the increment is 12.2 N, and $\tilde{q}^{\alpha}$ has the same value since this is the first increment. The largest residual force in the model, r^{α}_{max}, is −749 N, which is clearly greater than $0.005 \times \tilde{q}^{\alpha}$. r^{α}_{max} occurred at node 1051 in degree of freedom 1. Abaqus must also check for equilibrium of the moments in the model since this model includes shell elements. The moment/rotation field also failed to satisfy the equilibrium check.

Although failure to satisfy the equilibrium check is enough to cause Abaqus to try another iteration, you should also examine the displacement correction. In the first iteration of the first increment of the first step the largest increment of displacement, Δu^{α}_{max}, and the largest correction to displacement, c^{α}_{max}, are both -5.576×10^{-3} m; and the largest increment of rotation and correction to rotation are both 1.598×10^{-2} radians. Since the incremental values and the corrections are always equal in the first iteration of the first increment of the first step, the check that the largest corrections to nodal variables are less than 1% of the largest incremental values will always fail. However, if Abaqus judges the solution to be linear (a judgement based on the magnitude of the residuals, $r^{\alpha}_{max} < 10^{-8}\tilde{q}^{\alpha}$), it will ignore this criterion.

Since Abaqus did not find an equilibrium solution in the first iteration, it tries a second iteration, as shown below.

```
              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     2


 AVERAGE FORCE                          1.00       TIME AVG. FORCE        1.00
 LARGEST RESIDUAL FORCE               -0.173       AT NODE       1051   DOF  1
 LARGEST INCREMENT OF DISP.        -5.582E-03      AT NODE        651   DOF  3
 LARGEST CORRECTION TO DISP.       -7.050E-05      AT NODE       1201   DOF  1
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

 AVERAGE MOMENT                         1.12       TIME AVG. MOMENT       1.12
 LARGEST RESIDUAL MOMENT           -8.698E-04      AT NODE        208   DOF  5
 LARGEST INCREMENT OF ROTATION     -1.597E-02      AT NODE       1051   DOF  5
 LARGEST CORRECTION TO ROTATION     1.305E-04      AT NODE        409   DOF  4
          THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED
```

In the second iteration r^{α}_{max} has fallen to −0.173 N at node 1051 in degree of freedom 1. However, equilibrium is not satisfied in this iteration because $0.005 \times \tilde{q}^{\alpha}$, where $\tilde{q}^{\alpha}$ = 1.00 N, is still less than r^{α}_{max}. The displacement correction criterion also failed again because $c^{\alpha}_{max} = -7.050 \times 10^{-5}$, which occurred at node 1201 in degree of freedom 1, is more than 1% of $\Delta u^{\alpha}_{max} = -5.582 \times 10^{-3}$, the maximum displacement increment.

Both the moment residual check and the largest correction to rotation check were satisfied in this second iteration; however, Abaqus must perform two more iterations because the solutions did not pass the force residual check (or the largest correction to displacement criterion). The message file summaries for the additional iterations necessary to obtain a solution in the first increment are shown below.

```
              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     3

AVERAGE FORCE                       0.997       TIME AVG. FORCE       0.997
LARGEST RESIDUAL FORCE            -5.838E-03    AT NODE        459    DOF  2
LARGEST INCREMENT OF DISP.        -5.582E-03    AT NODE        651    DOF  3
LARGEST CORRECTION TO DISP.        9.150E-06    AT NODE        559    DOF  3
         FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

AVERAGE MOMENT                       1.12       TIME AVG. MOMENT      1.12
LARGEST RESIDUAL MOMENT           -1.338E-06    AT NODE        908    DOF  5
LARGEST INCREMENT OF ROTATION     -1.597E-02    AT NODE       1051    DOF  5
LARGEST CORRECTION TO ROTATION     3.233E-05    AT NODE        809    DOF  5
         THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED


              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     4

AVERAGE FORCE                       0.997       TIME AVG. FORCE       0.997
LARGEST RESIDUAL FORCE            -1.581E-07    AT NODE       1002    DOF  1
LARGEST INCREMENT OF DISP.        -5.582E-03    AT NODE        651    DOF  3
LARGEST CORRECTION TO DISP.        1.945E-09    AT NODE        559    DOF  3
         THE FORCE     EQUILIBRIUM EQUATIONS HAVE CONVERGED

AVERAGE MOMENT                       1.12       TIME AVG. MOMENT      1.12
LARGEST RESIDUAL MOMENT            3.691E-10    AT NODE        259    DOF  5
LARGEST INCREMENT OF ROTATION     -1.597E-02    AT NODE       1051    DOF  5
LARGEST CORRECTION TO ROTATION     6.461E-09    AT NODE        809    DOF  5
         THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED

ITERATION SUMMARY FOR THE INCREMENT:   4 TOTAL ITERATIONS, OF WHICH
  0 ARE SEVERE DISCONTINUITY ITERATIONS AND  4 ARE EQUILIBRIUM ITERATIONS.

TIME INCREMENT COMPLETED  0.100    ,  FRACTION OF STEP COMPLETED  0.100
STEP TIME COMPLETED       0.100    ,  TOTAL TIME COMPLETED        0.100
```

After four iterations $\tilde{q}^{\alpha} = 0.997$ N and $r^{\alpha}_{max} = -1.581 \times 10^{-7}$ N at node 1002 in degree of freedom 1. These values satisfy $r^{\alpha}_{max} < 0.005 \times \tilde{q}^{\alpha}$, so the force residual check is satisfied. Comparing c^{α}_{max} to the largest increment of displacement shows that the displacement correction is below the required tolerance. The solution for the forces and displacements has, therefore, converged. The checks for both the moment residual and the rotation correction continue to be satisfied, as they have been since the second iteration. With a solution that satisfies equilibrium for all variables (displacement and rotation in this case), the first load increment is complete. The increment summary shows the number of iterations that were required for this increment, the size of the increment, and the fraction of the step that has been completed.

The second increment requires two iterations to converge, as shown below.

```
 INCREMENT     2 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  0.100

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1

AVERAGE FORCE                        10.2       TIME AVG. FORCE       6.33
LARGEST RESIDUAL FORCE               -4.11      AT NODE        459    DOF  2
LARGEST INCREMENT OF DISP.        -5.585E-03    AT NODE        651    DOF  3
LARGEST CORRECTION TO DISP.        1.846E-04    AT NODE        509    DOF  3
         FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

AVERAGE MOMENT                       2.27       TIME AVG. MOMENT      1.70
LARGEST RESIDUAL MOMENT           -1.226E-02    AT NODE        208    DOF  4
LARGEST INCREMENT OF ROTATION     -1.586E-02    AT NODE       1051    DOF  4
```

```
LARGEST CORRECTION TO ROTATION    -7.332E-04   AT NODE        409   DOF  5
         MOMENT    EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     2

AVERAGE FORCE                        10.2       TIME AVG. FORCE        6.33
LARGEST RESIDUAL FORCE            -5.316E-04   AT NODE        359   DOF  2
LARGEST INCREMENT OF DISP.        -5.587E-03   AT NODE        651   DOF  3
LARGEST CORRECTION TO DISP.       -2.954E-06   AT NODE        459   DOF  3
         THE FORCE     EQUILIBRIUM EQUATIONS HAVE CONVERGED

AVERAGE MOMENT                       2.67       TIME AVG. MOMENT       1.90
LARGEST RESIDUAL MOMENT           4.569E-07   AT NODE        208   DOF  4
LARGEST INCREMENT OF ROTATION     -1.586E-02   AT NODE       1051   DOF  4
LARGEST CORRECTION TO ROTATION     1.028E-05   AT NODE        209   DOF  4
         THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED
         TIME INCREMENT MAY NOW INCREASE TO    0.150

ITERATION SUMMARY FOR THE INCREMENT:   2 TOTAL ITERATIONS, OF WHICH
  0 ARE SEVERE DISCONTINUITY ITERATIONS AND  2 ARE EQUILIBRIUM ITERATIONS.

TIME INCREMENT COMPLETED  0.100    ,  FRACTION OF STEP COMPLETED  0.200
STEP TIME COMPLETED       0.200    ,  TOTAL TIME COMPLETED        0.200
```

Abaqus continues this process of applying an increment of load then iterating to find a solution until it completes the whole analysis (or reaches the increment specified as the value of the INC parameter). In this analysis it required four more increments. Abaqus gives a summary at the end of the message file of how the analysis progressed and how many error and warning messages it issued. The summary for this analysis is shown below. An important item to check is how many iterations Abaqus uses. In this analysis it performed 15 iterations over the six increments: the model's system of equations was solved 15 times (i.e., 15 matrix decompositions), illustrating the increased computational expense of nonlinear analyses compared with linear simulations.

```
         THE ANALYSIS HAS BEEN COMPLETED


    ANALYSIS SUMMARY:
    TOTAL OF          6  INCREMENTS
                      0  CUTBACKS IN AUTOMATIC INCREMENTATION
                     15  ITERATIONS INCLUDING CONTACT ITERATIONS IF PRESENT
                     15  PASSES THROUGH THE EQUATION SOLVER OF WHICH
                     15  INVOLVE MATRIX DECOMPOSITION, INCLUDING
                      0  DECOMPOSITION(S) OF THE MASS MATRIX
                      1  REORDERING OF EQUATIONS TO MINIMIZE WAVEFRONT
                      0  ADDITIONAL RESIDUAL EVALUATIONS FOR LINE SEARCHES
                      0  ADDITIONAL OPERATOR EVALUATIONS FOR LINE SEARCHES
                      1  WARNING MESSAGES DURING USER INPUT PROCESSING
                      0  WARNING MESSAGES DURING ANALYSIS
                      0  ANALYSIS WARNINGS ARE NUMERICAL PROBLEM MESSAGES
                      0  ANALYSIS WARNINGS ARE NEGATIVE EIGENVALUE MESSAGES
                      0  ERROR MESSAGES


    JOB TIME SUMMARY
      USER TIME (SEC)       =  0.50000
      SYSTEM TIME (SEC)     =   0.0000
      TOTAL CPU TIME (SEC)  =  0.50000
      WALLCLOCK TIME (SEC)  =           2
```

You should always check this summary at the end of every Abaqus simulation. It tells you if the analysis job ran to completion (i.e., whether it terminated without a FORTRAN error), and it gives the number of error and warning messages Abaqus issued during the simulation. Always investigate any errors or warnings. All warnings and errors generated during the analysis are found in the message (**.msg**) file. Warnings issued "during user input processing" are found in the data (**.dat**) file.

Data file

The tables of displacements and reaction forces that you requested are in the data (**.dat**) file. The midspan deflections at the end of the step can be found near the end of the file.

```
   THE FOLLOWING TABLE IS PRINTED FOR NODES BELONGING TO NODE SET MIDSPAN

        NODE FOOT-    U1           U2           U3           UR1          UR2          UR3
             NOTE

         601      -1.2795E-04 -4.4921E-05 -1.0831E-02
         602      -1.2457E-04 -4.5147E-05 -1.0749E-02
         603      -1.2218E-04 -4.5645E-05 -1.0679E-02
         604      -1.2070E-04 -4.5966E-05 -1.0625E-02
         605      -1.1891E-04 -4.6602E-05 -1.0581E-02
         606      -1.1749E-04 -4.6822E-05 -1.0553E-02
         607      -1.1489E-04 -4.7487E-05 -1.0537E-02
         608      -1.1213E-04 -4.7541E-05 -1.0541E-02
         609      -1.0685E-04 -4.8026E-05 -1.0561E-02

 MAXIMUM          -1.0685E-04 -4.4921E-05 -1.0537E-02   0.000        0.000        0.000
 AT NODE                609          601          607           0            0            0

 MINIMUM          -1.2795E-04 -4.8026E-05 -1.0831E-02   0.000        0.000        0.000
 AT NODE                601          609          601           0            0            0
```

Compare these with the displacements from the linear analysis in Chapter 5, "Using Shell Elements." The maximum displacement at the midspan in this simulation is about 9% less than that predicted from the linear analysis. Including the nonlinear geometric effects in the simulation reduces the vertical deflection (U3) of the midspan of the plate.

Another difference between the two analyses is that in the nonlinear simulation there are nonzero deflections in the 1- and 2-directions. What effects make the in-plane displacements, U1 and U2, nonzero in the nonlinear analysis? Why is the vertical deflection of the plate less in the nonlinear analysis?

The plate deforms into a curved shape: a geometry change that is taken into account in the nonlinear simulation. As a consequence, membrane effects cause some of the load to be carried by membrane action rather than by bending alone. This makes the plate stiffer. In addition, the pressure loading, which is always normal to the plate's surface, starts to have a component in the 1- and 2-directions as the plate deforms. The nonlinear analysis includes the effects of this stiffening and the changing direction of the pressure. Neither of these effects is included in the linear simulation.

The differences between the linear and nonlinear simulations are sufficiently large to indicate that a linear simulation is not adequate for this plate under this particular loading condition.

For five degree of freedom shells, such as the S8R5 element used in this analysis, Abaqus does not output total rotations at the nodes.

8.4.4 Postprocessing

When you are in the directory containing the output database file **skew_nl.odb**, type the following command at the operating system prompt:

```
abaqus viewer odb=skew_nl
```

Showing the available frames

To begin this exercise, determine the available output frames (the increment intervals at which results were written to the output database).

To show the available frames:

1. From the main menu bar, select **Result→Step/Frame**.

   The **Step/Frame** dialog box appears.

   During the analysis Abaqus/Standard wrote field output results to the output database file every second increment, as was requested. Abaqus/Viewer displays the list of the available frames, as shown in Figure 8–12.

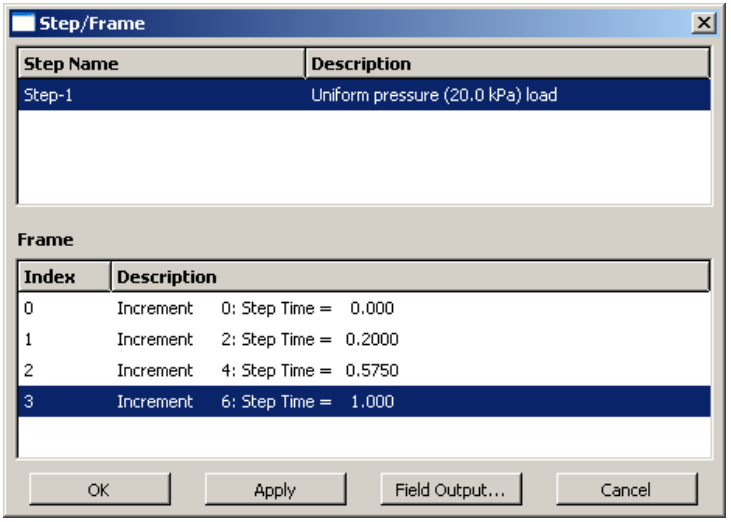


Figure 8–12 Available frames.

The list tabulates the steps and increments for which field variables are stored. This analysis consisted of a single step with six increments. The results for increment 0 (the initial state of the step) are saved by default, and you saved data for increments 2, 4, and 6. By default,

Abaqus/Viewer always uses the data for the last available increment saved in the output database file.

2. Click **OK** to dismiss the **Step/Frame** dialog box.

Displaying the deformed and undeformed model shapes

Use the **Allow Multiple Plot States** tool to display the deformed model shape with the undeformed model shape superimposed. Set the render style for both images to wireframe, and toggle off the translucency of the superimposed plot from the **Superimpose Plot Options** dialog box. Rotate the view to obtain a plot similar to that shown in Figure 8–13. By default, the deformed shape is plotted for the last increment. (For clarity, the edges of the undeformed shape are plotted using a dashed style.)

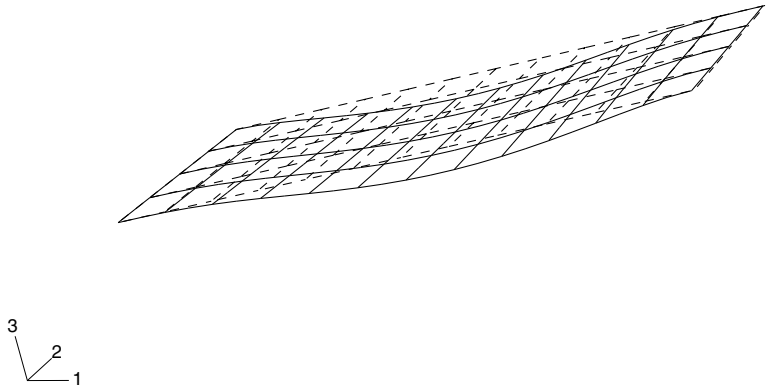


Figure 8–13 Deformed and undeformed model shapes of the skew plate.

Using results from other frames

You can evaluate the results from other increments saved to the output database file by selecting the appropriate frame.

To select a new frame:

1. From the main menu bar, select **Result→Step/Frame**.
   The **Step/Frame** dialog box appears.
2. Select `Increment 4` from the **Frame** menu.
3. Click **OK** to apply your changes and to close the **Step/Frame** dialog box.

Any plots now requested will use results from increment 4. Repeat this procedure, substituting the increment number of interest, to move through the output database file.

Note: Alternatively, you may use the **Frame Selector** dialog box to select a results frame.

X–Y plotting

You saved the displacements of the midspan nodes (node set **MIDSPAN**) in the history portion of the output database file **skew_nl.odb** for each increment of the simulation. You can use these results to create *X–Y* plots. In particular, you will plot the vertical displacement history of the nodes located at the edges of the plate midspan.

To create *X–Y* plots of the midspan displacements:

1. First, display only the nodes in the node set named **PART-1-1.MIDSPAN**: in the Results Tree, expand the **Node Sets** container underneath the output database file named **skew_nl.odb**. Click mouse button 3 on the set named **PART-1-1.MIDSPAN**, and select **Replace** from the menu that appears.
2. Use the **Common Plot Options** dialog box to show the node labels (i.e., numbers) to determine which nodes are located at the edges of the plate midspan.
3. In the Results Tree, expand the **History Output** container for the output database named **skew_nl.odb**.
4. Locate the output labeled as follows: **Spatial displacement: U3 at Node *xxx* in NSET MIDSPAN**. Each of these curves represents the vertical motion of one of the midspan nodes.

   Tip: Filter the container according to ***U3*** to facilitate your selection.
5. Select (using [Ctrl]+Click) the vertical motion of the two midspan edge nodes. Use the node labels to determine which curves you need to select.
6. Click mouse button 3, and select **Plot** from the menu that appears.

   Abaqus reads the data for both curves from the output database file and plots a graph similar to the one shown in Figure 8–14. (For clarity, the second curve has been changed to a dashed line, and the default grid and legend positions have been changed.)

   The nonlinear nature of this simulation is clearly seen in these curves: as the analysis progresses, the plate stiffens. In this simulation the increase in the plate stiffness with the deformation is due to membrane effects. Therefore, the resulting peak displacement is less than that predicted by the linear analysis, which did not include this effect.

You can create *X–Y* curves from either history or field data stored in the output database (**.odb**) file. *X–Y* curves can also be read from an external file or they can be typed into the Visualization module interactively. Once curves have been created, their data can be further manipulated and plotted to the screen in graphical form.

The *X–Y* plotting capability of Abaqus/Viewer is discussed further in Chapter 10, “Materials.”

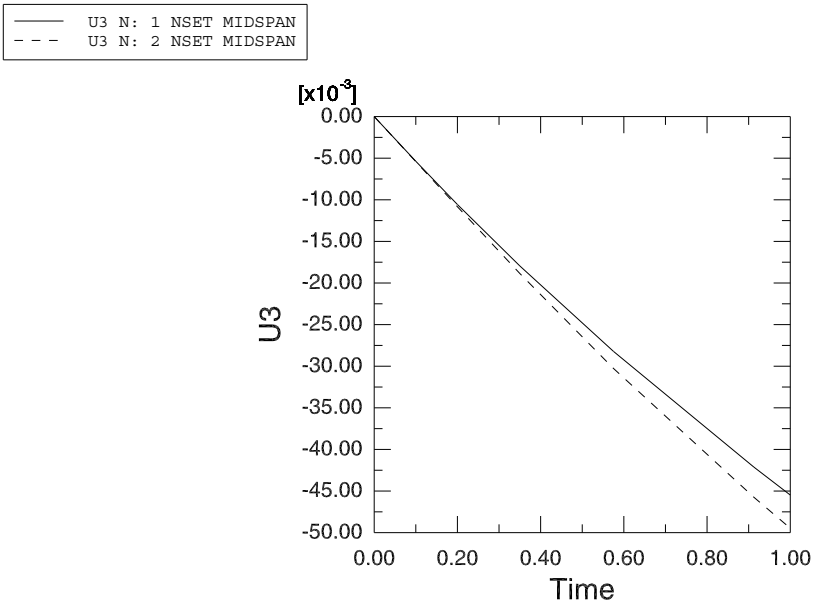


Figure 8–14 Midspan displacement history at the edges of the skew plate.

Tabular data

Create a tabular data report of the midspan displacements. Use the node set **`PART-1-1.MIDSPAN`** to create the appropriate display group and the frame selector to choose the final frame. The contents of the report are shown below.

```
Source 1
---------

   ODB: skew_nl.odb
   Step: Step-1
   Frame: Increment      6: Step Time =    1.000

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: PART-1-1

            Node            U.U1            U.U2            U.U3
           Label         @Loc 1          @Loc 1          @Loc 1
-----------------------------------------------------------------
             601    -2.68589E-03    -746.369E-06    -49.4577E-03
             602    -2.62498E-03    -749.228E-06    -48.9958E-03
             603    -2.57304E-03    -758.277E-06    -48.5853E-03
             604    -2.53788E-03    -761.475E-06    -48.1742E-03
             605    -2.48991E-03     -774.13E-06    -47.6904E-03
             606    -2.45666E-03    -777.171E-06    -47.1307E-03
             607    -2.40294E-03      -792.3E-06      -46.52E-03
             608    -2.36145E-03    -793.014E-06    -45.9489E-03
             609    -2.27792E-03    -805.258E-06    -45.4701E-03
```

```
Minimum            -2.68589E-03    -805.258E-06    -49.4577E-03

       At Node              601             609             601
Maximum            -2.27792E-03    -746.369E-06    -45.4701E-03

       At Node              609             601             609
```

8.4.5 Running the analysis in Abaqus/Explicit

As an optional exercise, you can modify the model and run a dynamic analysis of the skew plate in Abaqus/Explicit. To do so, you must add a density of 7800 kg/m^3 to the material definition and change the element type to S4R (with appropriate modifications to the element connectivity list). After making the appropriate model changes, you can create and run a new job to examine the transient dynamic effects in the plate under a suddenly applied load.

8.5 Related Abaqus examples

- "Elastic-plastic collapse of a thin-walled elbow under in-plane bending and internal pressure," Section 1.1.2 of the Abaqus Example Problems Manual
- "Laminated composite shells: buckling of a cylindrical panel with a circular hole," Section 1.2.2 of the Abaqus Example Problems Manual
- "Unstable static problem: reinforced plate under compressive loads," Section 1.2.5 of the Abaqus Example Problems Manual
- "Large rotation of a one degree of freedom system," Section 1.3.5 of the Abaqus Benchmarks Manual
- "Vibration of a cable under tension," Section 1.4.3 of the Abaqus Benchmarks Manual

8.6 Suggested reading

The following references provide additional information on nonlinear finite element methods. They allow the interested user to explore the topic in more depth.

General texts on nonlinear finite element analysis

- Belytschko, T., W. K. Liu, and B. Moran, *Nonlinear Finite Elements for Continua and Structures*, Wiley & Sons, 2000.
- Bonet, J., and R. D. Wood, *Nonlinear Continuum Mechanics for Finite Element Analysis*, Cambridge, 1997.
- Cook, R. D., D. S. Malkus, and M. E. Plesha, *Concepts and Applications of Finite Element Analysis*, Wiley & Sons, 1989.

- Crisfield, M. A., *Non-linear Finite Element Analysis of Solids and Structures, Volume I: Essentials*, Wiley & Sons, 1991.
- Crisfield, M. A., *Non-linear Finite Element Analysis of Solids and Structures, Volume II: Advanced Topics*, Wiley & Sons, 1997.
- E. Hinton (editor), *NAFEMS Introduction to Nonlinear Finite Element Analysis*, NAFEMS Ltd., 1992.
- Oden, J. T., *Finite Elements of Nonlinear Continua*, McGraw-Hill, 1972.

8.7 Summary

- There are three sources of nonlinearity in structural problems: material, geometric, and boundary (contact). Any combination of these may be present in an Abaqus analysis.
- Geometric nonlinearity occurs whenever the magnitude of the displacements affects the response of the structure. It includes the effects of large displacements and rotations, snap through, and load stiffening.
- In Abaqus/Standard nonlinear problems are solved iteratively using the Newton-Raphson method. A nonlinear problem will require many times the computer resources required by a linear problem.
- Abaqus/Explicit does not need to iterate to obtain a solution; however, the computational cost may be affected by reductions in the stable time increment due to large changes in geometry.
- A nonlinear analysis step is split into a number of increments.
  - Abaqus/Standard iterates to find the approximate static equilibrium obtained at the end of each new load increment. Abaqus/Standard controls the load incrementation by using convergence controls throughout the simulation.
  - Abaqus/Explicit determines a solution by advancing the kinematic state from one increment to the next, using a smaller time increment than what is commonly used in implicit analyses. The size of the increment is limited by the stable time increment. By default, time incrementation is completely automated in Abaqus/Explicit.
- The status file allows the progress of an analysis to be monitored while it is running. The message file contains the details of the load incrementation and iterations.
- Results can be saved at the end of each increment so that the evolution of the structure's response can be visualized in Abaqus/Viewer. Results can also be plotted in the form of *X–Y* graphs.

9. Nonlinear Explicit Dynamics

In previous chapters you explored the basics of explicit dynamics procedures; in this chapter you will examine this topic in greater detail. The explicit dynamics procedure can be an effective tool for solving a wide variety of nonlinear solid and structural mechanics problems. It is often complementary to an implicit solver such as Abaqus/Standard. From a user standpoint, the distinguishing characteristics of the explicit and implicit methods are:

- Explicit methods require a small time increment size that depends solely on the highest natural frequencies of the model and is independent of the type and duration of loading. Simulations generally take on the order of 10,000 to 1,000,000 increments, but the computational cost per increment is relatively small.
- Implicit methods do not place an inherent limitation on the time increment size; increment size is generally determined from accuracy and convergence considerations. Implicit simulations typically take orders of magnitude fewer increments than explicit simulations. However, since a global set of equations must be solved in each increment, the cost per increment of an implicit method is far greater than that of an explicit method.

Knowing these characteristics of the two procedures can help you decide which methodology is appropriate for your problems.

9.1 Types of problems suited for Abaqus/Explicit

Before discussing how the explicit dynamics procedure works, it is helpful to understand what classes of problems are well-suited to Abaqus/Explicit. Throughout this manual we have incorporated pertinent examples of the following classes of problems commonly performed in Abaqus/Explicit:

High-speed dynamic events

The explicit dynamics method was originally developed to analyze high-speed dynamic events that can be extremely costly to analyze using implicit programs, such as Abaqus/Standard. As an example of such a simulation, the effect of a short-duration blast load on a steel plate is analyzed in Chapter 10, “Materials.” Since the load is applied rapidly and is very severe, the response of the structure changes rapidly. Accurate tracking of stress waves through the plate is important for capturing the dynamic response. Since stress waves are associated with the highest frequencies of the system, obtaining an accurate solution requires many small time increments.

Complex contact problems

Contact conditions are formulated more easily using an explicit dynamics method than using an implicit method. The result is that Abaqus/Explicit can readily analyze problems involving complex contact interaction between many independent bodies. Abaqus/Explicit is particularly well-suited for analyzing the transient dynamic response of structures that are subject to impact loads and

subsequently undergo complex contact interaction within the structure. An example of such a problem is the circuit board drop test presented in Chapter 12, "Contact." In this example a circuit board sitting in foam packaging is dropped on the floor from a height of 1 m. The problem involves impact between the packaging and the floor, as well as rapidly changing contact conditions between the circuit board and the packaging.

Complex postbuckling problems

Unstable postbuckling problems are solved readily in Abaqus/Explicit. In such problems the stiffness of the structure changes drastically as the loads are applied. Postbuckling response often includes the effects of contact interactions.

Highly nonlinear quasi-static problems

For a variety of reasons Abaqus/Explicit is often very efficient in solving certain classes of problems that are essentially static. Quasi-static process simulation problems involving complex contact such as forging, rolling, and sheet-forming generally fall within these classes. Sheet forming problems usually include very large membrane deformations, wrinkling, and complex frictional contact conditions. Bulk forming problems are characterized by large distortions, flash formation, and contact interaction with the dies. An example of a quasi-static forming simulation is presented in Chapter 13, "Quasi-Static Analysis with Abaqus/Explicit."

Materials with degradation and failure

Material degradation and failure often lead to severe convergence difficulties in implicit analysis programs, but Abaqus/Explicit models such materials well. An example of material degradation is the concrete cracking model, in which tensile cracking causes the material stiffness to become negative. An example of material failure is the ductile failure model for metals, in which material stiffness can degrade until it reduces to zero. At this time the failed elements are removed from the model entirely.

Each of these types of analyses can include temperature and heat transfer effects.

9.2 Explicit dynamic finite element methods

This section contains an algorithmic description of the Abaqus/Explicit solver as well as a comparison between implicit and explicit time integration and a discussion of the advantages of the explicit dynamics method.

9.2.1 Explicit time integration

Abaqus/Explicit uses a central difference rule to integrate the equations of motion explicitly through time, using the kinematic conditions at one increment to calculate the kinematic conditions at the next

increment. At the beginning of the increment the program solves for dynamic equilibrium, which states that the nodal mass matrix, $\mathbf{M}$, times the nodal accelerations, $\ddot{\mathbf{u}}$, equals the net nodal forces (the difference between the external applied forces, $\mathbf{P}$, and internal element forces, $\mathbf{I}$):

$$\mathbf{M}\ddot{\mathbf{u}} = \mathbf{P} - \mathbf{I}.$$

The accelerations at the beginning of the current increment (time t) are calculated as

$$\ddot{\mathbf{u}}|_{(t)} = (\mathbf{M})^{-1} \cdot (\mathbf{P} - \mathbf{I})\,|_{(t)}.$$

Since the explicit procedure always uses a diagonal, or lumped, mass matrix, solving for the accelerations is trivial; there are no simultaneous equations to solve. The acceleration of any node is determined completely by its mass and the net force acting on it, making the nodal calculations very inexpensive.

The accelerations are integrated through time using the central difference rule, which calculates the change in velocity assuming that the acceleration is constant. This change in velocity is added to the velocity from the middle of the previous increment to determine the velocities at the middle of the current increment:

$$\dot{\mathbf{u}}|_{\left(t+\frac{\Delta t}{2}\right)} = \dot{\mathbf{u}}|_{\left(t-\frac{\Delta t}{2}\right)} + \frac{\left(\Delta t|_{(t+\Delta t)} + \Delta t|_{(t)}\right)}{2}\ \ddot{\mathbf{u}}|_{(t)}.$$

The velocities are integrated through time and added to the displacements at the beginning of the increment to determine the displacements at the end of the increment:

$$\mathbf{u}|_{(t+\Delta t)} = \mathbf{u}|_{(t)} + \Delta t|_{(t+\Delta t)}\dot{\mathbf{u}}|_{\left(t+\frac{\Delta t}{2}\right)}.$$

Thus, satisfying dynamic equilibrium at the beginning of the increment provides the accelerations. Knowing the accelerations, the velocities and displacements are advanced "explicitly" through time. The term "explicit" refers to the fact that the state at the end of the increment is based solely on the displacements, velocities, and accelerations at the beginning of the increment. This method integrates constant accelerations exactly. For the method to produce accurate results, the time increments must be quite small so that the accelerations are nearly constant during an increment. Since the time increments must be small, analyses typically require many thousands of increments. Fortunately, each increment is inexpensive because there are no simultaneous equations to solve. Most of the computational expense lies in the element calculations to determine the internal forces of the elements acting on the nodes. The element calculations include determining element strains and applying material constitutive relationships (the element stiffness) to determine element stresses and, consequently, internal forces.

Here is a summary of the explicit dynamics algorithm:

1. Nodal calculations.
   a. Dynamic equilibrium.

$$\ddot{\mathbf{u}}_{(t)} = \mathbf{M}^{-1}\left(\mathbf{P}_{(t)} - \mathbf{I}_{(t)}\right)$$

b. Integrate explicitly through time.

$$\dot{\mathbf{u}}_{\left(t+\frac{\Delta t}{2}\right)} = \dot{\mathbf{u}}_{\left(t-\frac{\Delta t}{2}\right)} + \frac{\left(\Delta t_{(t+\Delta t)} + \Delta t_{(t)}\right)}{2} \quad \ddot{\mathbf{u}}_t$$

$$\mathbf{u}_{(t+\Delta t)} = \mathbf{u}_{(t)} + \Delta t_{(t+\Delta t)} \dot{\mathbf{u}}_{\left(t+\frac{\Delta t}{2}\right)}.$$

2. Element calculations.
   a. Compute element strain increments, $\Delta\boldsymbol{\varepsilon}$, from the strain rate, $\dot{\boldsymbol{\varepsilon}}$.
   b. Compute stresses, $\boldsymbol{\sigma}$, from constitutive equations.

$$\boldsymbol{\sigma}_{(t+\Delta t)} = f\left(\boldsymbol{\sigma}_{(t)}, \Delta\boldsymbol{\varepsilon}\right)$$

   c. Assemble nodal internal forces, $\mathbf{I}_{(t+\Delta t)}$.
3. Set $t + \Delta t$ to t and return to Step 1.

9.2.2 Comparison of implicit and explicit time integration procedures

For both the implicit and the explicit time integration procedures, equilibrium is defined in terms of the external applied forces, $\mathbf{P}$, the internal element forces, $\mathbf{I}$, and the nodal accelerations:

$$\mathbf{M}\ddot{\mathbf{u}} = \mathbf{P} - \mathbf{I},$$

where $\mathbf{M}$ is the mass matrix. Both procedures solve for nodal accelerations and use the same element calculations to determine the internal element forces. The biggest difference between the two procedures lies in the manner in which the nodal accelerations are computed. In the implicit procedure a set of linear equations is solved by a direct solution method. The computational cost of solving this set of equations is high when compared to the relatively low cost of the nodal calculations with the explicit method.

Abaqus/Standard uses automatic incrementation based on the full Newton iterative solution method. Newton's method seeks to satisfy dynamic equilibrium at the end of the increment at time $t + \Delta t$ and to compute displacements at the same time. The time increment, Δt, is relatively large compared to that used in the explicit method because the implicit scheme is unconditionally stable. For a nonlinear problem each increment typically requires several iterations to obtain a solution within the prescribed tolerances. Each Newton iteration solves for a correction, $\mathbf{c}_j$, to the incremental displacements, $\Delta\mathbf{u}_j$. Each iteration requires the solution of a set of simultaneous equations,

$$\widehat{\mathbf{K}}_j \mathbf{c}_j = \mathbf{P}_j - \mathbf{I}_j - \mathbf{M}_j \ddot{\mathbf{u}}_j,$$

which is an expensive procedure for large models. The effective stiffness matrix, $\widehat{\mathbf{K}}_j$, is a linear combination of the tangent stiffness matrix and the mass matrix for the iteration. The iterations continue until several quantities—force residual, displacement correction, etc.—are within the prescribed tolerances. For a smooth nonlinear response Newton's method has a quadratic rate of convergence, as illustrated below:

Iteration	Relative Error
1	1
2	10^{-2}
3	10^{-4}
.	.
.	.
.	.

However, if the model contains highly discontinuous processes, such as contact and frictional sliding, quadratic convergence may be lost and a large number of iterations may be required. Cutbacks in the time increment size may become necessary to satisfy equilibrium. In extreme cases the resulting time increment size in the implicit analysis may be on the same order as a typical stable time increment for an explicit analysis, while still carrying the high solution cost of implicit iteration. In some cases convergence may not be possible using the implicit method.

Each iteration in an implicit analysis requires solving a large system of linear equations, a procedure that requires considerable computation, disk space, and memory. For large problems these equation solver requirements are dominant over the requirements of the element and material calculations, which are similar for an analysis in Abaqus/Explicit. As the problem size increases, the equation solver requirements grow rapidly so that, in practice, the maximum size of an implicit analysis that can be solved on a given machine often is dictated by the amount of disk space and memory available on the machine rather than by the required computation time.

9.2.3 Advantages of the explicit time integration method

The explicit method is especially well-suited to solving high-speed dynamic events that require many small increments to obtain a high-resolution solution. If the duration of the event is short, the solution can be obtained efficiently.

Contact conditions and other extremely discontinuous events are readily formulated in the explicit method and can be enforced on a node-by-node basis without iteration. The nodal accelerations can be adjusted to balance the external and internal forces during contact.

The most striking feature of the explicit method is the absence of a global tangent stiffness matrix, which is required with implicit methods. Since the state of the model is advanced explicitly, iterations and tolerances are not required.

9.3 Automatic time incrementation and stability

The stability limit dictates the maximum time increment used by the Abaqus/Explicit solver. It is a critical factor in the performance of Abaqus/Explicit. The following sections describe the stability limit and discuss how Abaqus/Explicit determines this value. Issues surrounding the model design parameters that affect the stability limit are also addressed. These model parameters include the model mass, material, and mesh.

9.3.1 Conditional stability of the explicit method

With the explicit method the state of the model is advanced through an increment of time, Δt, based on the state of the model at the start of the increment at time t. The amount of time that the state can be advanced and still remain an accurate representation of the problem is typically quite short. If the time increment is larger than this maximum amount of time, the increment is said to have exceeded the *stability limit*. A possible effect of exceeding the stability limit is a numerical instability, which may lead to an unbounded solution. It generally is not possible to determine the stability limit exactly, so conservative estimates are used instead. The stability limit has a great effect on reliability and accuracy, so it must be determined consistently and conservatively. For computational efficiency Abaqus/Explicit chooses the time increments to be as close as possible to the stability limit without exceeding it.

9.3.2 Definition of the stability limit

The stability limit is defined in terms of the highest frequency in the system ($\omega_{\max}$). Without damping the stability limit is defined by the expression

$$\Delta t_{\text{stable}} = \frac{2}{\omega_{\max}},$$

and with damping it is defined by the expression

$$\Delta t_{\text{stable}} = \frac{2}{\omega_{\max}} \left(\sqrt{1+\xi^2} - \xi \right),$$

where ξ is the fraction of critical damping in the mode with the highest frequency. (Recall that critical damping defines the limit between oscillatory and non-oscillatory motion in the context of free-damped vibration. Abaqus/Explicit always introduces a small amount of damping in the form of bulk viscosity to control high-frequency oscillations.) Perhaps contrary to engineering intuition, damping always reduces the stability limit.

The actual highest frequency in the system is based on a complex set of interacting factors, and it is not computationally feasible to calculate its exact value. Alternately, we use a simple estimate that is

efficient and conservative. Instead of looking at the global model, we estimate the highest frequency of each individual element in the model, which is always associated with the dilatational mode. It can be shown that the highest element frequency determined on an element-by-element basis is always higher than the highest frequency in the assembled finite element model.

Based on the element-by-element estimate, the stability limit can be redefined using the element length, L^e, and the wave speed of the material, c_d:

$$\Delta t_{\text{stable}} = \frac{L^e}{c_d}.$$

For most element types—a distorted quadrilateral element, for example—the above equation is only an estimate of the actual element-by-element stability limit because it is not clear how the element length should be determined. As an approximation the shortest element distance can be used, but the resulting estimate is not always conservative. The shorter the element length, the smaller the stability limit. The wave speed is a property of the material. For a linear elastic material with a Poisson's ratio of zero

$$c_d = \sqrt{\frac{E}{\rho}},$$

where E is Young's modulus and ρ is the mass density. The stiffer the material, the higher the wave speed, resulting in a smaller stability limit. The higher the density, the lower the wave speed, resulting in a larger stability limit.

Our simplified stability limit definition provides some intuitive understanding. The stability limit is the transit time of a dilatational wave across the distance defined by the characteristic element length. If we know the size of the smallest element dimension and the wave speed of the material, we can estimate the stability limit. For example, if the smallest element dimension is 5 mm and the dilatational wave speed is 5000 m/s, the stable time increment is on the order of 1×10^{-6} s.

9.3.3 Fully automatic time incrementation versus fixed time incrementation in Abaqus/Explicit

Abaqus/Explicit uses equations such as those discussed in the previous section to adjust the time increment size throughout the analysis so that the stability limit, based on the current state of the model, is never exceeded. Time incrementation is automatic and requires no user intervention, not even a suggested initial time increment. The stability limit is a mathematical concept resulting from the numerical model. Since the finite element program has all of the relevant details, it can determine an efficient and conservative stability limit. However, Abaqus/Explicit does allow the user to override the automatic time incrementation, if desired.

The time increment used in an explicit analysis must be smaller than the stability limit of the central-difference operator. Failure to use a small enough time increment will result in an unstable solution. When the solution becomes unstable, the time history response of solution variables such

as displacements will usually oscillate with increasing amplitudes. The total energy balance will also change significantly. If the model contains only one material type, the initial time increment is directly proportional to the size of the smallest element in the mesh. If the mesh contains uniform size elements but contains multiple material descriptions, the element with the highest wave speed will determine the initial time increment.

In nonlinear problems—those with large deformations and/or nonlinear material response—the highest frequency of the model will continually change, which consequently changes the stability limit. Abaqus/Explicit has two strategies for time incrementation control: fully automatic time incrementation (where the code accounts for changes in the stability limit) and fixed time incrementation.

Two types of estimates are used to determine the stability limit: element by element and global. An analysis always starts by using the element-by-element estimation method and may switch to the global estimation method under certain circumstances.

The element-by-element estimate is conservative; it will give a smaller stable time increment than the true stability limit that is based upon the maximum frequency of the entire model. In general, constraints such as boundary conditions and kinematic contact have the effect of compressing the eigenvalue spectrum, and the element-by-element estimates do not take this into account.

The adaptive, global estimation algorithm determines the maximum frequency of the entire model using the current dilatational wave speed. This algorithm continuously updates the estimate for the maximum frequency. The global estimator will usually allow time increments that exceed the element-by-element values.

A fixed time incrementation scheme is also available in Abaqus/Explicit. The fixed time increment size is determined either by the initial element-by-element stability estimate for the step or by a time increment specified directly by the user. Fixed time incrementation may be useful when a more accurate representation of the higher mode response of a problem is required. In this case, a time increment size smaller than the element-by-element estimates may be used. When fixed time incrementation is used, Abaqus/Explicit will not check that the computed response is stable during the step. The user should ensure that a valid response has been obtained by carefully checking the energy history and other response variables.

9.3.4 Mass scaling to control time incrementation

Since the mass density influences the stability limit, under some circumstances scaling the mass density can potentially increase the efficiency of an analysis. For example, because of the complex discretization of many models, there are often regions containing very small or poorly shaped elements that control the stability limit. These controlling elements are often few in number and may exist in localized areas. By increasing the mass of only these controlling elements, the stability limit can be increased significantly, while the effect on the overall dynamic behavior of the model may be negligible.

The automatic mass scaling features in Abaqus/Explicit can keep offending elements from hindering the stability limit. There are two fundamental approaches used in mass scaling: defining a scaling factor directly or defining a desired element-by-element stable time increment for the elements whose mass is to be scaled. These two approaches, described in detail in “Mass scaling,” Section 11.6.1 of the Abaqus

Analysis User's Manual, permit additional user control over the stability limit. However, use caution when employing mass scaling since significantly changing the mass of the model may change the physics of the problem.

9.3.5 Effect of material on stability limit

The material model affects the stability limit through its effect on the dilatational wave speed. In a linear material the wave speed is constant; therefore, the only changes in the stability limit during the analysis result from changes in the smallest element dimension during the analysis. In a nonlinear material, such as a metal with plasticity, the wave speed changes as the material yields and the stiffness of the material changes. Abaqus/Explicit monitors the effective wave speeds in the model throughout the analysis, and the current material state in each element is used for stability estimates. After yielding, the stiffness decreases, reducing the wave speed and, consequently, increasing the stability limit.

9.3.6 Effect of mesh on stability limit

Since the stability limit is roughly proportional to the shortest element dimension, it is advantageous to keep the element size as large as possible. Unfortunately, for accurate analyses a fine mesh is often necessary. To obtain the highest possible stability limit while using the required level of mesh refinement, the best approach is to have a mesh that is as uniform as possible. Since the stability limit is based on the smallest element dimension in the model, even a single small or poorly shaped element can reduce the stability limit drastically. For diagnostic purposes Abaqus/Explicit provides a list in the status (`.sta`) file of the 10 elements in the mesh with the lowest stability limit. If the model contains some elements whose stability limits are much lower than those of the rest of the mesh, remeshing the model more uniformly may be worthwhile.

9.3.7 Numerical instability

Abaqus/Explicit remains stable for most elements under most circumstances. It is possible, however, to define spring and dashpot elements such that they become unstable during the course of an analysis. Therefore, it is useful to be able to recognize a numerical instability if it occurs in your analysis. If it does occur, the result typically will be unbounded, nonphysical, and often characterized by oscillatory solutions.

9.4 Example: stress wave propagation in a bar

This example demonstrates some of the fundamental ideas in explicit dynamics described earlier in Chapter 2, "Abaqus Basics." It also illustrates stability limits and the effect of mesh refinement and material properties on the solution time.

The bar has the dimensions shown in Figure 9–1.

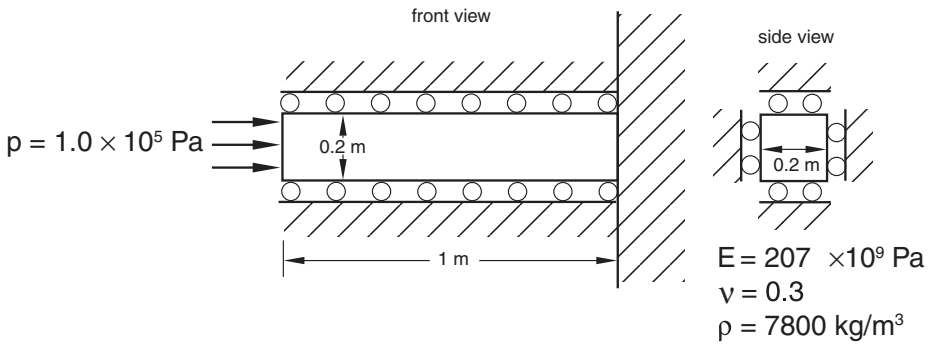


Figure 9–1 Schematic for wave propagation in a bar.

To make the problem a one-dimensional strain problem, all four lateral faces are on rollers; thus, the three-dimensional model simulates a one-dimensional problem. The material is steel with the properties shown in Figure 9–1. The free end of the bar is subjected to a blast load with a magnitude of 1.0×10^5 Pa and a duration of 3.88×10^{-5} s. The normalized load versus time is shown in Figure 9–2.

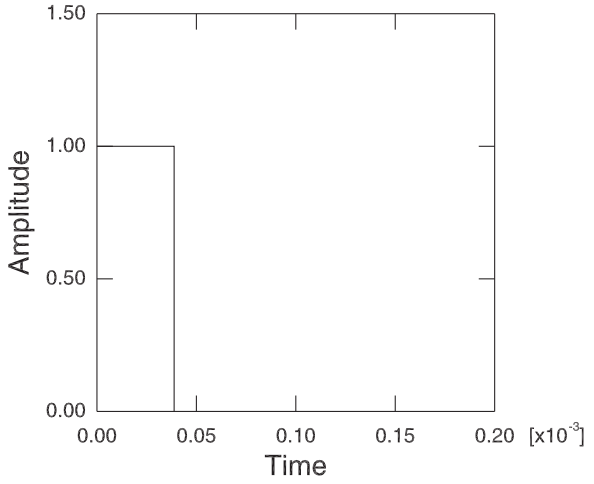


Figure 9–2 Blast amplitude versus time.

Using the material properties (neglecting Poisson's ratio), we can calculate the wave speed of the material using the equations introduced in the previous section.

$$c_d = \sqrt{\frac{E}{\rho}} = \sqrt{\frac{207 \times 10^9 \text{ Pa}}{7800 \text{ kg/m}^3}} = 5.15 \times 10^3 \text{ m/s}.$$

At this speed the wave passes to the fixed end of the bar in 1.94×10^{-4} s. Since we are interested in the stress propagation along the length of the bar through time, we need an adequately refined mesh to capture the stress wave accurately. We will assume that the blast load will take place over the span of 10 elements. To determine the length of these 10 elements, multiply the blast duration by the wave speed:

$$L_{10el} = \left(3.88 \times 10^{-5} \text{ s}\right) c_d = 0.2 \text{ m}.$$

The length of 10 elements is 0.2 m. Since the total length of the bar is 1.0 m, we would have 50 elements along the length. To keep the mesh uniform, we will also have 10 elements in each of the transverse directions, making the mesh 50 × 10 × 10. This mesh is shown in Figure 9–3.

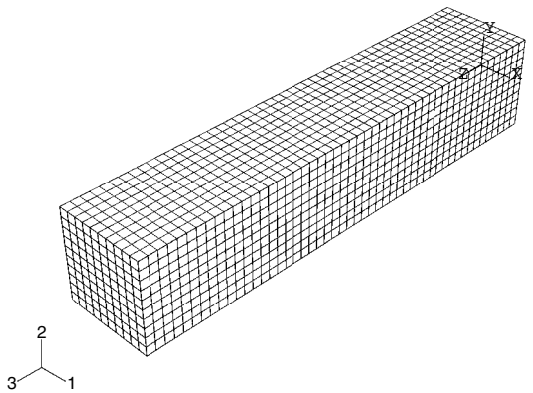


Figure 9–3 50 × 10 × 10 mesh.

Create this mesh in your preprocessor. Use the coordinate system shown in Figure 9–3.

9.4.1 Node and element sets

This example defines node and element sets to apply the loads and boundary conditions and to visualize output. The node sets are defined on their respective faces, as shown in Figure 9–4.

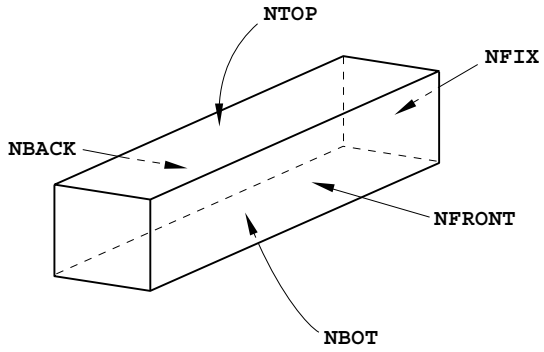


Figure 9–4 Node sets.

The element sets are defined as shown in Figure 9–5.

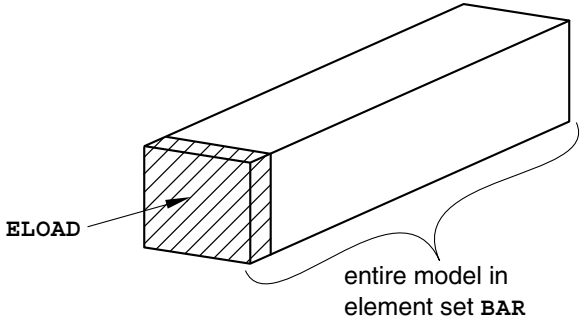


Figure 9–5 Element sets for modeling.

In addition, this example defined an element set containing three elements in the center of the bar. You can define this element set manually by selecting these elements such that their faces nearest to the free end are at distances 0.25 m, 0.5 m, and 0.75 m from the free end, as shown in Figure 9–6. These elements will be used for postprocessing.

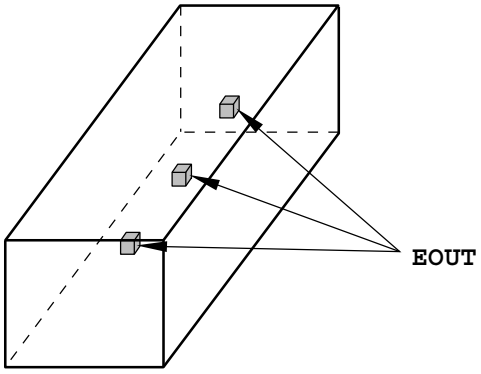


Figure 9–6 Element sets for postprocessing.

9.4.2 Reviewing the input file—the model data

In this section you will review your input file and include additional information.

Model description

The following would be a suitable description in the *HEADING option for this simulation:

```
*HEADING
Stress wave propagation in a bar -- 50x10x10 elements
SI units (kg, m, s, N)
```

Element connectivity

If you create input files using a preprocessor, check to make sure that you are using the correct element type (C3D8R). It is possible that the preprocessor specified the element type incorrectly. The *ELEMENT option block in this model begins with the following:

```
*ELEMENT, TYPE=C3D8R, ELSET=BAR
```

If you created this input file using a preprocessor, the name given for the ELSET parameter in your model may not be **BAR**. If necessary, change the name to **BAR**.

Section properties

The section properties are the same for all elements. In the following option statement, the element set **BAR** is used to assign the material properties to the elements.

```
*SOLID SECTION, ELSET=BAR, MATERIAL=STEEL
```

Material properties

The bar is made of steel, which we assume to be linear elastic with a Young's modulus of 207×10^9 Pa, a Poisson's ratio of 0.3, and a density of 7800 kg/m^3. The following material option block specifies these values:

```
*MATERIAL, NAME=STEEL
*ELASTIC
207.0E9, 0.3
*DENSITY
7800.0,
```

Fixed boundary conditions

In this model, we fix all the translations at the built-in, right-hand end of the bar, then constrain the front, back, top, and bottom faces of the bar so that these faces are on rollers and the strain is uniaxial. Using the node sets defined previously, the following boundary conditions are used in this model:

```
*BOUNDARY
NFIX, 1, 3
NFRONT, 1, 1
NBACK, 1, 1
NTOP, 2, 2
NBOT, 2, 2
```

Amplitude definition

The blast load is applied at its maximum value instantaneously and is held constant for 3.88×10^{-5} s. Then the load is suddenly removed and held constant at zero. The *AMPLITUDE option is used to define the time variation of loads and boundary conditions. On the data lines following the *AMPLITUDE option, pairs of data are given in the form:

<time>, <amplitude>, <time>, <amplitude>, etc.

Up to four data pairs can be entered on each data line. Abaqus considers the amplitude to be held constant following the last amplitude value given. The following *AMPLITUDE option block defines the amplitude for the blast load:

```
*AMPLITUDE, NAME=BLAST
0., 1., 3.88E-5, 1., 3.89E-5, 0, 3.90E-5, 0.
```

9.4.3 Reviewing the input file—the history data

We will now review the history data associated with this problem, including the step definition, loading, bulk viscosity, and output requests.

Step definition

The step definition indicates that this is an explicit dynamics analysis with a duration of 2.0×10^{-4} s. You can also include a descriptive title for the step.

```
*STEP
Blast loading
*DYNAMIC, EXPLICIT
, 2.0E-4
```

Loading

Apply the pressure load with a value of 1.0×10^5 Pa to the free face of the bar, which you previously defined to be in an element set called **ELOAD**. The pressure load at any given time is the magnitude specified under the *DLOAD option times the value interpolated from the amplitude curve. To apply the load correctly, you need to determine the face identifier label of the free element faces. For the model defined in "Stress wave propagation in a bar," Section A.7, the free face is face number 3, which corresponds to the pressure identifier P3. The face identifier depends on the order in which the nodes are defined on the *ELEMENT option, as shown in Figure 9–7. Use the amplitude named **BLAST** when applying the pressure load.

```
*DLOAD, AMPLITUDE=BLAST
ELOAD, <P1, P2, P3, P4, P5, or P6>, 1.0E5
```

If you define the pressure load in your preprocessor, the correct face identifier should be determined automatically.

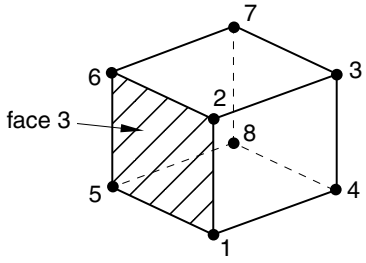


Figure 9–7 Face label identifier for a C3D8R element.

Bulk viscosity

To keep the stress wave as sharp as possible, the quadratic bulk viscosity (discussed in "Bulk viscosity," Section 9.5.1) is set to zero.

```
*BULK VISCOSITY
0.06, 0.0
```

Output requests

By default, many preprocessors create an Abaqus input file that has a large number of output request options. If you created your input file using a preprocessor and you find that these default output options were created, delete them all because they will generally generate too much output.

You want to have an output database file created during the analysis so that you can use Abaqus/Viewer to postprocess the results. Four output database frames (intervals at which data are written to the output database) are adequate to show the stress wave propagating through the mesh. This example sets the parameter VARIABLE=PRESELECT on the *OUTPUT, FIELD option to write the default field data for a *DYNAMIC, EXPLICIT procedure to the output database file. In addition, stress (S) history output in element set **EOUT** is requested for every increment.

```
*OUTPUT, FIELD, VARIABLE=PRESELECT, NUMBER INTERVAL=4
*OUTPUT, HISTORY, FREQUENCY=1
*ELEMENT OUTPUT, ELSET=EOUT
S,
*END STEP
```

9.4.4 Running the analysis

After storing your input in a file called **wave_50x10x10.inp**, run the analysis using the following command:

```
abaqus job=wave_50x10x10
```

If your analysis does not complete, check the data file, **wave_50x10x10.dat**, and status file, **wave_50x10x10.sta**, for error messages. Modify your input file to remove the errors. If you still have trouble running your analysis, compare your input file to the one given in "Stress wave propagation in a bar," Section A.7.

Status file

The status file, **wave_50x10x10.sta**, contains information about moments of inertia, followed by information concerning the initial stability limit. The 10 elements with the lowest stable time limits are listed in rank order.

```
Most critical elements:
 Element number   Rank    Time increment   Increment ratio
-----------------------------------------------------------
         1           1       1.819458E-06      1.000000E+00
        19           2       1.819458E-06      1.000000E+00
       201           3       1.819458E-06      1.000000E+00
       219           4       1.819458E-06      1.000000E+00
       301           5       1.819458E-06      1.000000E+00
       319           6       1.819458E-06      1.000000E+00
       501           7       1.819458E-06      1.000000E+00
       519           8       1.819458E-06      1.000000E+00
       601           9       1.819458E-06      1.000000E+00
       619          10       1.819458E-06      1.000000E+00
```

The status file continues with information about the progress of the solution.

```
 STEP 1  ORIGIN 0.0000

  Total memory used for step 1 is approximately 6.9 megabytes.
  Global time estimation algorithm will be used.
  Scaling factor:  1.0000
  Variable mass scaling factor at zero increment:  1.0000

              STEP      TOTAL       CPU       STABLE    CRITICAL    KINETIC      TOTAL
INCREMENT     TIME       TIME      TIME    INCREMENT     ELEMENT     ENERGY     ENERGY
        0  0.000E+00 0.000E+00  00:00:00 1.819E-06           1  0.000E+00  0.000E+00
Results number 0 at increment zero.
ODB Field Frame Number      0 of      4 requested intervals at increment zero.
ODB Field Frame Number      0 of      2 requested intervals at increment zero.
        6  1.092E-05 1.092E-05  00:00:00 1.819E-06           1  4.432E-05 -1.134E-06
       12  2.183E-05 2.183E-05  00:00:00 1.819E-06         201  9.043E-05 -1.437E-06
       17  3.318E-05 3.318E-05  00:00:00 2.896E-06           1  1.376E-04 -1.201E-06
       21  4.474E-05 4.474E-05  00:00:00 2.882E-06           1  1.547E-04  1.875E-06
       23  5.050E-05 5.050E-05  00:00:00 2.877E-06           1  1.533E-04  2.685E-06
ODB Field Frame Number      1 of      4 requested intervals at  5.049687E-05
       27  6.199E-05 6.199E-05  00:00:00 2.870E-06           1  1.517E-04  9.508E-07
.
.
.
```

9.4.5 Postprocessing

Start Abaqus/Viewer by typing the following command at the operating system prompt:

```
abaqus viewer odb=wave_50x10x10
```

Plotting the stress along a path

We are interested in looking at how the stress distribution along the length of the bar changes with time. To do so, we will look at the stress distribution at three different times throughout the course of the analysis.

Create a curve of the variation of the stress in the 3-direction (S33) along the axis of the bar for each of the first three frames of the output database file. To create these plots, you first need to define a straight path along the axis of the bar.

To create a point list path along the center of the bar:

1. In the Results Tree, double-click **Paths**.

   The **Create Path** dialog box appears.

2. Name the path `Center`. Select **Point list** as the path type, and click **Continue**.

   The **Edit Point List Path** dialog box appears.

3. In the **Point Coordinates** table, enter the coordinates of the centers of both ends of the bar. The input specifies a path from the first point to the second point, as defined in the global coordinate system of the model.

   Note: If you generated the geometry and mesh using the procedure described earlier, the table entries are `0, 0, 1` and `0, 0, 0`. If you used an alternate procedure to generate the bar

geometry, you can use the tool in the **Query** toolbar to determine the coordinates of the centers at each end of the bar.

4. When you have finished, click **OK** to close the **Edit Point List Path** dialog box.

To save *X–Y* plots of stress along the path at three different times:

1. In the Results Tree, double-click **XYData**.

   The **Create XY Data** dialog box appears.

2. Choose **Path** as the *X–Y* data source, and click **Continue**.

   The **XY Data from Path** dialog box appears with the path that you created visible in the list of available paths. If the undeformed model shape is currently displayed, the path you select is highlighted in the plot.

3. Toggle on **Include intersections** under **Point Locations**.

4. Accept **True distance** as the selection in the **X Values** region of the dialog box.

5. Click **Field Output** in the **Y Values** region of the dialog box to open the **Field Output** dialog box.

6. Select the S33 stress component, and click **OK**.

   The field output variable in the **XY Data from Path** dialog box changes to indicate that stress data in the 3-direction (S33) will be created.

   Note: Abaqus/Viewer may warn you that the field output variable will not affect the current image. Leave the plot state **As is**, and click **OK** to continue.

7. Click **Step/Frame** in the **Y Values** region of the **XY Data from Path** dialog box.

8. In the **Step/Frame** dialog box that appears, choose frame 1, which is the second of the five recorded frames. (The first frame listed, frame 0, is the base state of the model at the beginning of the step.) Click **OK**.

   The **Y Values** region of the **XY Data from Path** dialog box changes to indicate that data from Step 1, frame 1 will be created.

9. To save the *X–Y* data, click **Save As**.

   The **Save XY Data As** dialog box appears.

10. Name the *X–Y* data **`S33_T1`**, and click **OK**.

    `S33_T1` appears in the **XYData** container of the Results Tree.

11. Repeat Steps 7 through 9 to create *X–Y* data for frames 2 and 3. Name the data sets **`S33_T2`** and **`S33_T3`**, respectively.

12. To close the **XY Data from Path** dialog box, click **Cancel**.

To plot the stress curves:

1. In the **XYData** container, drag the cursor to select all three *X*–*Y* data sets.
2. Click mouse button 3, and select **Plot** from the menu that appears.
   Abaqus/Viewer plots the stress in the 3-direction along the center of the bar for frames 1, 2, and 3, corresponding to approximate simulation times of 5×10^{-5} s, 1×10^{-4} s, and 1.5×10^{-4} s, respectively.
3. Click in the prompt area to cancel the current procedure.

To customize the *X*–*Y* plot:

1. Double-click the *Y*-axis.
   The **Axis Options** dialog box appears. The **Y Axis** is selected.
2. In the **Tick Mode** region of the **Scale** tabbed page, select **By increment** and specify that the *Y*-axis major tick marks occur at `20E3` Pa increments.
   You can also customize the axis titles.
3. Switch to the **Title** tabbed page.
4. Enter `Stress - S33 (Pa)` as the *Y*-axis title.
5. To edit the *X*-axis, select the axis label in the **X Axis** field of the dialog box. In the **Title** tabbed page of the dialog box, enter `Distance along bar (m)` as the *X*-axis title.
6. Click **Dismiss** to close the **Axis Options** dialog box.

To customize the appearance of the curves in the *X*–*Y* plot:

1. In the Visualization toolbox, click to open the **Curve Options** dialog box.
2. In the **Curves** field, select `S33_T2`.
3. Choose the dotted line style for the `S33_T2` curve.
   The `S33_T2` curve becomes dotted.
4. Repeat Steps 2 and 3 to make the `S33_T3` curve dashed.
5. Dismiss the **Curve Options** dialog box.
   The customized plot appears in Figure 9–8. (For clarity, the default grid and legend positions have been changed.)

We can see that the length of the bar affected by the stress wave is approximately 0.2 m in each of the three curves. This distance should correspond to the distance that the blast wave travels during its time of application, which can be checked by a simple calculation. If the length of the wave front is 0.2 m and the wave speed is 5.15×10^3 m/s, the time it takes for the wave to travel 0.2 m is 3.88×10^{-5} s.

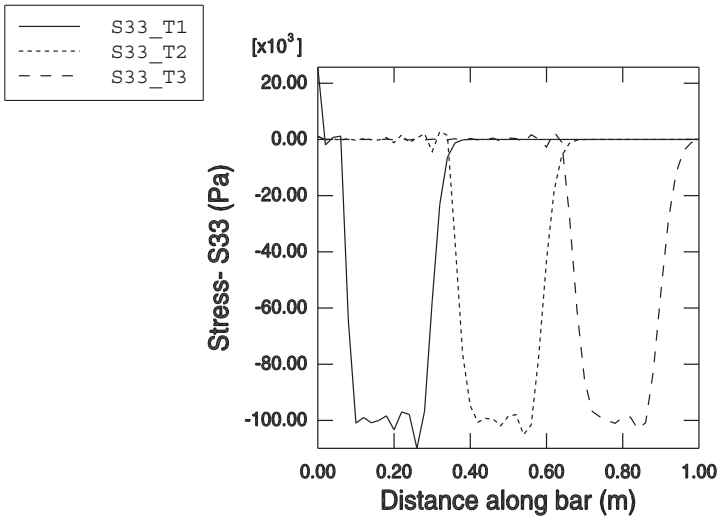


Figure 9–8 Stress (S33) along the bar at three different time instances.

As expected, this is the duration of the blast load that we applied. The stress wave is not exactly square as it passes along the bar. In particular, there is "ringing" or oscillation of the stress behind the sudden changes in stress. Linear bulk viscosity, discussed later in this chapter, damps the ringing so that it does not affect the results adversely.

Creating a history plot

Another way to study the results is to view the time history of stress at three different points within the bar.

To plot the stress history:

1. In the Results Tree, click mouse button 3 on **History Output** and deselect **Group Children** from the menu that appears.
2. Select the data for the three elements. Use [Ctrl]+Click to select multiple *X*–*Y* data sets.
3. Click mouse button 3, and select **Plot** from the menu that appears.

   Abaqus/Viewer displays an *X*–*Y* plot of the longitudinal stress in each element versus time.
4. Click in the prompt area to cancel the current procedure.

   As before, you can customize the appearance of the plot.

To customize the *X–Y* plot:

1. Double-click the *X*-axis.
   The **Axis Options** dialog box appears.
2. Switch to the **Title** tabbed page.
3. Specify `Total time (s)` as the *X*-axis title.
4. Click **Dismiss** to close the dialog box.

To customize the appearance of the curves in the *X–Y* plot:

1. In the Visualization toolbox, click to open the **Curve Options** dialog box.
2. In the **Curves** field, select the temporary *X–Y* data label that corresponds to the element closest to the free end of the bar. (Of the elements in this set, this one is affected first by the stress wave.)
3. Enter `S33-0.25` as the curve legend text.
4. In the **Curves** field, select the temporary *X–Y* data label that corresponds to the element in the middle of the bar. (This is the element affected next by the stress wave.)
5. Specify `S33-0.5` as the curve legend text, and change the curve style to dotted.
6. In the **Curves** field, select the temporary *X–Y* data label that corresponds to the element closest to the fixed end of the bar. (This is the element affected last by the stress wave.)
7. Specify `S33-0.75` as the curve legend text, and change the curve style to dashed.
8. Click **Dismiss** to close the dialog box.
   The customized plot appears in Figure 9–9. (For clarity, the default grid and legend positions have been changed.)

In the history plot we can see that stress at a given point increases as the stress wave travels through the point. Once the stress wave has passed completely through the point, the stress at the point oscillates about zero.

9.4.6 How the mesh affects the stable time increment and CPU time

In "Automatic time incrementation and stability," Section 9.3, we discussed how mesh refinement affects the stability limit and the CPU time. Here we will illustrate this effect with the wave propagation problem. We began with a reasonably refined mesh of square elements with 50 elements along the length and 10 elements in each of the two transverse directions. For illustrative purposes, we will now use a coarse mesh of 25 × 5 × 5 elements and observe how refining the mesh in the various directions changes the CPU time. The four meshes are shown in Figure 9–10.

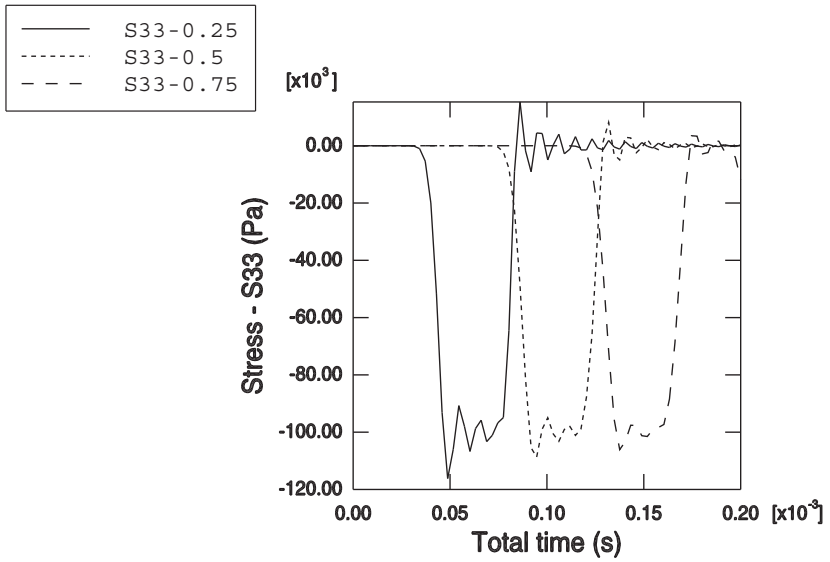


Figure 9–9 Time history of stress (S33) at three points along the length of the bar (0.25 m, 0.5 m, and 0.75 m).

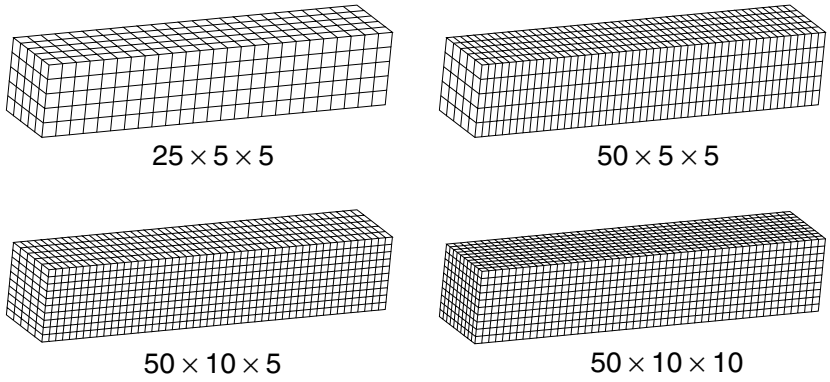


Figure 9–10 Meshes from least to most refined.

Table 9–1 shows how the CPU time (normalized with respect to the coarse mesh model result) changes with mesh refinement for this problem. The first half of the table provides the expected results, based on the simplified stability equations presented in this guide; the second half of the table provides the results obtained by running the analyses in Abaqus/Explicit on a desktop workstation.

Table 9–1 Mesh refinement and solution time.

Mesh	Simplified Theory			Actual		
	Δt_{stable} (s)	Number of Elements	CPU Time (s)	Max Δt_{stable} (s)	Number of Elements	Normalized CPU Time
25 × 5 × 5	A	B	C	6.06e-6	625	1
50 × 5 × 5	A/2	2B	4C	3.14e-6	1250	4
50 × 10 × 5	A/2	4B	8C	3.12e-6	2500	8.33
50 × 10 × 10	A/2	8B	16C	3.11e-6	5000	16.67

For the theoretical results we choose the coarsest mesh, 25 × 5 × 5, as the base state, and we define the stable time increment, the number of elements, and the CPU time as variables A, B, and C, respectively. As the mesh is refined, two things happen: the smallest element dimension decreases, and the number of elements in the mesh increases. Each of these effects increases the CPU time. In the first level of refinement, the 50 × 5 × 5 mesh, the smallest element dimension is cut in half and the number of elements is doubled, increasing the CPU time by a factor of four over the previous mesh. However, further doubling the mesh to 50 × 10 × 5 does not change the smallest element dimension; it only doubles the number of elements. Therefore, the CPU time increases by only a factor of two over the 50 × 5 × 5 mesh. Further refining the mesh so that the elements are uniform and square in the 50 × 10 × 10 mesh again doubles the number of elements and the CPU time.

This simplified calculation predicts quite well the trends of how mesh refinement affects the stable time increment and CPU time. However, there are reasons why we did not compare the predicted and actual stable time increment values. First, recall that we made the approximation that the stable time increment is

$$\Delta t_{\text{stable}} = \frac{L^e}{c_d}.$$

We then assumed that the characteristic element length, L^e, is the smallest element dimension, whereas Abaqus/Explicit actually determines the characteristic element length based on the overall size and shape of the element. Another complication is that Abaqus/Explicit employs a global stability estimator, which allows a larger stable time increment to be used. These factors make it difficult to predict the stable time increment accurately before running the analysis. However, since the trends follow nicely from the simplified theory, it is straightforward to predict how the stable time increment will change with mesh refinement.

9.4.7 How the material affects the stable time increment and CPU time

The same wave propagation analysis performed on different materials would take different amounts of CPU time, depending on the wave speed of the material. For example, if we were to change the material from steel to aluminum, the wave speed would change from 5.15×10^3 m/s to

$$c_d = \sqrt{\frac{E}{\rho}} = \sqrt{\frac{70 \times 10^9 \text{ Pa}}{2700 \text{ kg/m}^3}} = 5.09 \times 10^3 \text{ m/s}.$$

The change from aluminum to steel has minimal effect on the stable time increment, because the stiffness and the density differ by nearly the same amount. In the case of lead the difference is more substantial, as the wave speed decreases to

$$c_d = \sqrt{\frac{E}{\rho}} = \sqrt{\frac{14 \times 10^9 \text{ Pa}}{11240 \text{ kg/m}^3}} = 1.12 \times 10^3 \text{ m/s},$$

which is approximately one-fifth the wave speed of steel. The stable time increment for the lead bar would be five times the stable time increment of our steel bar.

9.5 Damping of dynamic oscillations

There are two reasons for adding damping to a model: to limit numerical oscillations or to add physical damping to the system. Abaqus/Explicit provides several methods of introducing damping into the analysis.

9.5.1 Bulk viscosity

Bulk viscosity introduces damping associated with volumetric straining. Its purpose is to improve the modeling of high-speed dynamic events. Abaqus/Explicit contains linear and quadratic forms of bulk viscosity. You can set bulk viscosity to nondefault values from step to step by using the *BULK VISCOSITY option, although it is rarely necessary to do so. The bulk viscosity pressure is not included in the material point stresses because it is intended as a numerical effect only. As such, it is not considered part of the material's constitutive response.

Linear bulk viscosity

By default, linear bulk viscosity is always included to damp "ringing" in the highest element frequency. It generates a bulk viscosity pressure that is linear in the volumetric strain rate, according to the following equation:

$$p_1 = b_1 \rho c_d L^e \dot{\varepsilon}_{vol},$$

where b_1 is a damping coefficient, whose default value is 0.06, ρ is the current material density, c_d is the current dilatational wave speed, L^e is the element characteristic length, and $\dot{\varepsilon}_{vol}$ is the volumetric strain rate.

Quadratic bulk viscosity

Quadratic bulk viscosity is included only in continuum elements (except for the plane stress element, CPS4R) and is applied only if the volumetric strain rate is compressive. The bulk viscosity pressure is quadratic in the strain rate, according to the following equation:

$$p_2 = \rho \left(b_2 L^e\right)^2 |\dot{\varepsilon}_{vol}| \, min\left(0, \dot{\varepsilon}_{vol}\right),$$

where b_2 is the damping coefficient, whose default value is 1.2.

The quadratic bulk viscosity smears a shock front across several elements and is introduced to prevent elements from collapsing under extremely high velocity gradients. Consider a simple one-element problem in which the nodes on one side of the element are fixed and the nodes on the other side have an initial velocity in the direction of the fixed nodes, as shown in Figure 9–11.

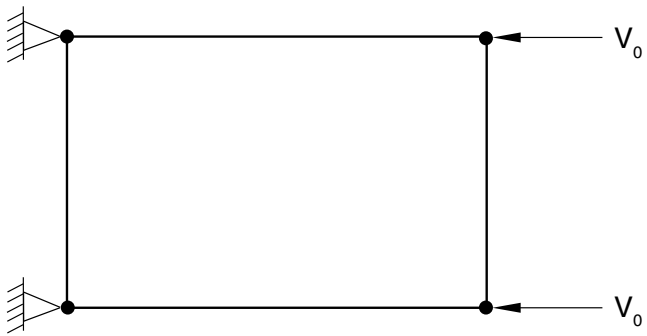


Figure 9–11 Element with fixed nodes and prescribed velocities.

The stable time increment size is precisely the transit time of a dilatational wave across the element. Therefore, if the initial nodal velocity is equal to the dilatational wave speed of the material, the element collapses to zero volume in one time increment. The quadratic bulk viscosity pressure introduces a resisting pressure that prevents the element from collapsing.

Fraction of critical damping due to bulk viscosity

The bulk viscosity pressures are based on only the dilatational modes of each element. The fraction of critical damping in the highest element mode is given by the following equation:

$$\xi = b_1 - b_2^2 \frac{L^e}{c_d} min\left(0, \dot{\varepsilon}_{vol}\right),$$

where ξ is the fraction of critical damping. The linear term alone represents 6% of critical damping, whereas the quadratic term is usually much smaller.

9.5.2 Viscous pressure

Viscous pressure loads are commonly used in structural problems and quasi-static problems to damp out the low-frequency dynamic effects, thus allowing static equilibrium to be reached in a minimal number of increments. These loads are applied as distributed loads (∗DLOAD) defined by the following formula:

$$p = -c_v \left(\bar{v} \cdot \bar{n} \right),$$

where p is the pressure applied to the body; c_v is the viscosity, given on the data line as the magnitude of the load; $\bar{v}$ is the velocity vector of the point on the surface where the viscous pressure is being applied; and $\bar{n}$ is the unit outward normal vector to the surface at the same point. For typical structural problems it is not desirable to absorb all of the energy. Typically, c_v is set equal to a small percentage—perhaps 1 or 2 percent—of the quantity ρc_d as an effective way of minimizing ongoing dynamic effects.

9.5.3 Material damping

The material model itself may provide damping in the form of plastic dissipation or viscoelasticity. For many applications such damping may be adequate. Another option is to use Rayleigh damping defined using the ∗DAMPING option, which is part of the ∗MATERIAL option block. There are two damping factors associated with Rayleigh damping: α_R for mass proportional damping and β_R for stiffness proportional damping.

Mass proportional damping

The α_R factor defines a damping contribution proportional to the mass matrix for an element. The damping forces that are introduced are caused by the absolute velocities of nodes in the model. The resulting effect can be likened to the model moving through a viscous fluid so that any motion of any point in the model triggers damping forces. Reasonable mass proportional damping does not reduce the stability limit significantly.

Stiffness proportional damping

The β_R factor defines damping proportional to the elastic material stiffness. A "damping stress," σ_d, proportional to the total strain rate is introduced, using the following formula:

$$\tilde{\sigma}_d = \beta_R \tilde{D}^{el} \dot{\varepsilon},$$

where $\dot{\varepsilon}$ is the strain rate. For hyperelastic and hyperfoam materials $\tilde{D}^{el}$ is defined as the initial elastic stiffness. For all other materials $\tilde{D}^{el}$ is the material's current elastic stiffness. This damping stress is added to the stress caused by the constitutive response at the integration point when the dynamic equilibrium equations are formed, but it is not included in the stress output. Damping can be introduced for any nonlinear analysis and provides standard Rayleigh damping for linear analyses. For a linear analysis stiffness proportional damping is exactly the same as defining a

damping matrix equal to β_R times the stiffness matrix. Stiffness proportional damping must be used with caution because it may significantly reduce the stability limit. To avoid a dramatic drop in the stable time increment, the stiffness proportional damping factor, β_R, should be less than or of the same order of magnitude as the initial stable time increment without damping.

9.5.4 Discrete dashpots

Yet another option is to define individual dashpot elements. Each dashpot element provides a damping force proportional to the relative velocity of its two nodes. The advantage of this approach is that it enables you to apply damping only at points where you decide it is necessary. Dashpots always should be used in parallel with other elements, such as springs or trusses, so that they do not cause a significant reduction in the stability limit.

9.6 Energy balance

Energy output is often an important part of an Abaqus/Explicit analysis. Comparisons between various energy components can be used to help evaluate whether an analysis is yielding an appropriate response.

9.6.1 Statement of energy balance

An energy balance for the entire model can be written as

$$E_I + E_V + E_{FD} + E_{KE} + E_{IHE} - E_W - E_{PW} - E_{CW} - E_{MW} - E_{HF} = E_{total} = \text{constant},$$

where E_I is the internal energy, E_V is the viscous energy dissipated, E_{FD} is the frictional energy dissipated, E_{KE} is the kinetic energy, E_{IHE} is the internal heat energy, E_W is the work done by the externally applied loads, and E_{PW}, E_{CW}, and E_{MW} are the work done by contact penalties, by constraint penalties, and by propelling added mass, respectively. E_{HF} is the external heat energy through external fluxes. The sum of these energy components is E_{total}, which should be constant. In the numerical model E_{total} is only approximately constant, generally with an error of less than 1%.

Internal energy

The internal energy is the sum of the recoverable elastic strain energy, E_E; the energy dissipated through inelastic processes such as plasticity, E_P; the energy dissipated through viscoelasticity or creep, E_{CD}; the artificial strain energy, E_A; the energy dissipated through damage, E_{DMD}; the energy dissipated through distortion control, E_{DC}; and the fluid cavity energy, E_{FC}:

$$E_I = E_E + E_P + E_{CD} + E_A + E_{DMD} + E_{DC} + E_{FC}.$$

The artificial strain energy includes energy stored in hourglass resistances and transverse shear in shell and beam elements. Large values of artificial strain energy indicate that mesh refinement or other changes to the mesh are necessary.

Viscous energy

The viscous energy is the energy dissipated by damping mechanisms, including bulk viscosity damping and material damping. A fundamental variable in the global energy balance, viscous energy is not part of the energy dissipated through viscoelasticity or inelastic processes.

External work of applied forces

The external work is integrated forward continuously, defined entirely by nodal forces (moments) and displacements (rotations). Prescribed boundary conditions also contribute to the external work.

9.6.2 Output of the energy balance

Each of the energy quantities can be requested as output and can be plotted as time histories summed over the entire model, particular element sets, individual elements, or as energy density within each element. The variable names associated with the energy quantities summed over the entire model or element sets are as listed in Table 9–2.

Table 9–2 Whole model energy output variables.

Variable Name	Energy Quantity
ALLIE	Internal energy, E_I: ALLIE = ALLSE + ALLPD + ALLCD + ALLAE + ALLDMD + ALLDC + ALLFC.
ALLKE	Kinetic energy, E_{KE}.
ALLVD	Viscous dissipated energy, E_V.
ALLFD	Frictional dissipated energy, E_{FD}.
ALLCD	Energy dissipated by viscoelasticity, E_{CD}.
ALLWK	Work of the external forces, E_W.
ALLPW	Work done by contact penalties, E_{PW}.
ALLCW	Work done by constraint penalties, E_{CW}.
ALLMW	Work done by propelling added mass (due to mass scaling), E_{MW}.
ALLSE	Elastic strain energy, E_E.
ALLPD	Inelastic dissipated energy, E_P.
ALLAE	Artificial strain energy, E_A.
ALLIHE	Internal heat energy, E_{IHE}.
ALLHF	External heat energy through external fluxes, E_{HF}.
ALLDMD	Energy dissipated by damage, E_{DMD}.

Variable Name	Energy Quantity
ALLDC	Energy dissipated by distortion control, E_{DC}.
ALLFC	Fluid cavity energy (negative of work done by fluid cavities), E_{FC}.
ETOTAL	Energy balance: $E_{TOT} = E_I + E_V + E_{FD} + E_{KE} + E_{IHE} - E_W - E_{PW} - E_{CW} - E_{MW} - E_{HF}$.

In addition, Abaqus/Explicit can produce element-level energy output and energy density output, as listed in Table 9–3.

Table 9–3 Whole element energy output variables.

Variable Name	Whole Element Energy Quantity
ELSE	Elastic strain energy.
ELPD	Plastic dissipated energy.
ELCD	Creep dissipated energy.
ELVD	Viscous dissipated energy.
ELASE	Artificial energy = drill energy + hourglass energy.
EKEDEN	Kinetic energy density in the element.
ESEDEN	Elastic strain energy density in the element.
EPDDEN	Plastic energy density dissipated in the element.
EASEDEN	Artificial strain energy density in the element.
ECDDEN	Creep strain energy density dissipated in the element.
EVDDEN	Viscous energy density dissipated in the element.
ELDMD	Energy dissipated in the element by damage.

9.7 Summary

- Abaqus/Explicit uses a central difference rule to integrate the kinematics explicitly through time.
- The explicit method requires many small time increments. Since there are no simultaneous equations to solve, each increment is inexpensive.
- The explicit method has great cost savings over the implicit method as the model size increases.
- The *stability limit* is the maximum time increment that can be used to advance the kinematic state and still remain accurate.

- Abaqus/Explicit automatically controls the time increment size throughout the analysis to maintain stability.
- As the material stiffness increases, the stability limit decreases; as the material density increases, the stability limit increases.
- For a mesh with a single material, the stability limit is roughly proportional to the smallest element dimension.
- Generally, mass proportional damping is used in Abaqus/Explicit to damp low-frequency oscillations, and stiffness proportional damping is used to damp high-frequency oscillations.
- In some situations an Abaqus/Explicit analysis may become unstable. The example problems in this chapter describe how to recognize and rectify instabilities.

10. Materials

The material library in Abaqus allows most engineering materials to be modeled, including metals, plastics, rubbers, foams, composites, granular soils, rocks, and plain and reinforced concrete. This guide discusses only three of the most commonly used material models: linear elasticity, metal plasticity, and rubber elasticity. All of the material models are discussed in detail in Part V, "Materials," of the Abaqus Analysis User's Manual.

10.1 Defining materials in Abaqus

You can use any number of different materials in your simulation. Each material definition starts with a *MATERIAL option. The NAME parameter identifies the name associated with the material being defined. This name is used to assign the material definition to specific elements in the model.

The material definition is one of the few situations in which the position of option blocks in the Abaqus input file is important. All of the option blocks defining specific aspects of a material's behavior, such as its elastic modulus or density, must follow the *MATERIAL option directly. Furthermore, the material option blocks defining the behavior of a particular material cannot be interrupted by other nonmaterial options. Abaqus issues an error message if it cannot associate a material behavior option block, such as *ELASTIC, with a prior *MATERIAL option.

For example, consider a material description, such as an elastic-plastic metal subjected to gravitational loads, that requires several material behavior option blocks to supply Abaqus with the necessary data. In addition to the elastic and plastic property option blocks, Abaqus needs the material's density to calculate the gravitational loads. Thus, the complete material description would be

```
*MATERIAL, NAME=STEEL
*ELASTIC            <-- Elastic properties
2.1E11, 0.3
*PLASTIC    <-- Plastic properties
2.0E8, 0.0
3.0E8, 0.2
*DENSITY    <---- Density
7800.0,
```

A non-material option block between the *PLASTIC and *DENSITY options, as shown in the following input, would cause Abaqus to terminate the analysis with an error message.

```
*MATERIAL, NAME=STEEL
*ELASTIC
2.1E11, 0.3
*PLASTIC
2.0E8, 0.0
3.0E8, 0.2
*BOUNDARY
101, 1, 3,
*DENSITY
7800.0,
```

Because of this option block Abaqus does not know which *MATERIAL option this option block belongs to.

10.2 Plasticity in ductile metals

Many metals have approximately linear elastic behavior at low strain magnitudes (see Figure 10–1), and the stiffness of the material, known as the Young's or elastic modulus, is constant.

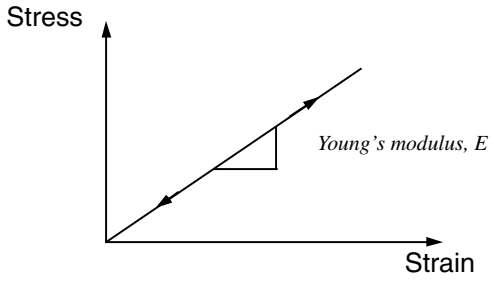


Figure 10–1 Stress-strain behavior for a linear elastic material, such as steel, at small strains.

At higher stress (and strain) magnitudes, metals begin to have nonlinear, inelastic behavior (see Figure 10–2), which is referred to as plasticity.

10.2.1 Characteristics of plasticity in ductile metals

The plastic behavior of a material is described by its yield point and its post-yield hardening. The shift from elastic to plastic behavior occurs at a certain point, known as the elastic limit or yield point, on a material's stress-strain curve (see Figure 10–2). The stress at the yield point is called the yield stress. In most metals the initial yield stress is 0.05 to 0.1% of the material's elastic modulus.

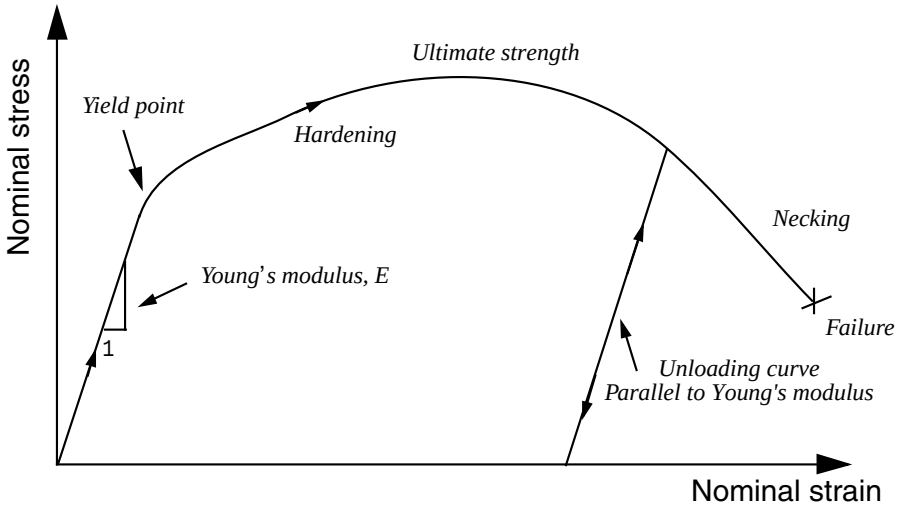


Figure 10–2 Nominal stress-strain behavior of an elastic-plastic material in a tensile test.

The deformation of the metal prior to reaching the yield point creates only elastic strains, which are fully recovered if the applied load is removed. However, once the stress in the metal exceeds the yield stress, permanent (inelastic) deformation begins to occur. The strains associated with this permanent deformation are called plastic strains. Both elastic and plastic strains accumulate as the metal deforms in the post-yield region.

The stiffness of a metal typically decreases dramatically once the material yields (see Figure 10–2). A ductile metal that has yielded will recover its initial elastic stiffness when the applied load is removed (see Figure 10–2). Often the plastic deformation of the material increases its yield stress for subsequent loadings: this behavior is called work hardening.

Another important feature of metal plasticity is that the inelastic deformation is associated with nearly incompressible material behavior. Modeling this effect places some severe restrictions on the type of elements that can be used in elastic-plastic simulations.

A metal deforming plastically under a tensile load may experience highly localized extension and thinning, called *necking*, as the material fails (see Figure 10–2). The engineering stress (force per unit undeformed area) in the metal is known as the *nominal stress*, with the conjugate *nominal strain* (length change per unit undeformed length). The nominal stress in the metal as it is necking is much lower than the material's ultimate strength. This material behavior is caused by the geometry of the test specimen, the nature of the test itself, and the stress and strain measures used. For example, testing the same material in compression produces a stress-strain plot that does not have a necking region because the specimen is not going to thin as it deforms under compressive loads. A mathematical model describing the plastic behavior of metals should be able to account for differences in the compressive and tensile behavior independent of the structure's geometry or the nature of the applied loads. This goal can be accomplished

if the familiar definitions of nominal stress, F/A_0, and nominal strain, $\Delta l/l_0$, where the subscript 0 indicates a value from the undeformed state of the material, are replaced by new measures of stress and strain that account for the change in area during the finite deformations.

10.2.2 Stress and strain measures for finite deformations

Strains in compression and tension are the same only if considered in the limit as $\Delta l \to dl \to 0$; i.e.,

$$d\varepsilon = \frac{dl}{l}$$

and

$$\varepsilon = \int_{l_0}^{l} \frac{dl}{l} = \ln\left(\frac{l}{l_0}\right),$$

where l is the current length, l_0 is the original length, and ε is the *true strain* or *logarithmic strain*.

The stress measure that is the conjugate to the true strain is called the *true stress* and is defined as

$$\sigma = \frac{F}{A},$$

where F is the force in the material and A is the current area. A ductile metal subjected to finite deformations will have the same stress-strain behavior in tension and compression if true stress is plotted against true strain.

10.2.3 Defining plasticity in Abaqus

When defining plasticity data in Abaqus, you must use *true stress* and *true strain*. Abaqus requires these values to interpret the data correctly.

Quite often material test data are supplied using values of nominal stress and strain. In such situations you must use the expressions presented below to convert the plastic material data from nominal stress-strain values to true stress-strain values.

The relationship between true strain and nominal strain is established by expressing the nominal strain as

$$\varepsilon_{nom} = \frac{l - l_0}{l_0} = \frac{l}{l_0} - \frac{l_0}{l_0} = \frac{l}{l_0} - 1.$$

Adding unity to both sides of this expression and taking the natural log of both sides provides the relationship between the true strain and the nominal strain:

$$\varepsilon = \ln(1 + \varepsilon_{nom}).$$

The relationship between true stress and nominal stress is formed by considering the incompressible nature of the plastic deformation and assuming the elasticity is also incompressible, so

$$l_0 A_0 = lA.$$

The current area is related to the original area by

$$A = A_0 \frac{l_0}{l}.$$

Substituting this definition of A into the definition of true stress gives

$$\sigma = \frac{F}{A} = \frac{F}{A_0}\frac{l}{l_0} = \sigma_{nom}\left(\frac{l}{l_0}\right),$$

where

$$\frac{l}{l_0}$$

can also be written as

$$1 + \varepsilon_{nom}.$$

Making this final substitution provides the relationship between true stress and nominal stress and strain:

$$\sigma = \sigma_{nom}(1 + \varepsilon_{nom}).$$

These relationships are valid only prior to necking.

The *PLASTIC option in Abaqus defines the post-yield behavior for most metals. Abaqus approximates the smooth stress-strain behavior of the material with a series of straight lines joining the given data points. Any number of points can be used to approximate the actual material behavior; therefore, it is possible to use a very close approximation of the actual material behavior. The data on the *PLASTIC option define the true yield stress of the material as a function of true plastic strain. The first piece of data given defines the initial yield stress of the material and, therefore, should have a plastic strain value of zero.

The strains provided in material test data used to define the plastic behavior are not likely to be the plastic strains in the material. Instead, they will probably be the total strains in the material. You must decompose these total strain values into the elastic and plastic strain components. The plastic strain is obtained by subtracting the elastic strain, defined as the value of true stress divided by the Young's modulus, from the value of total strain (see Figure 10–3).

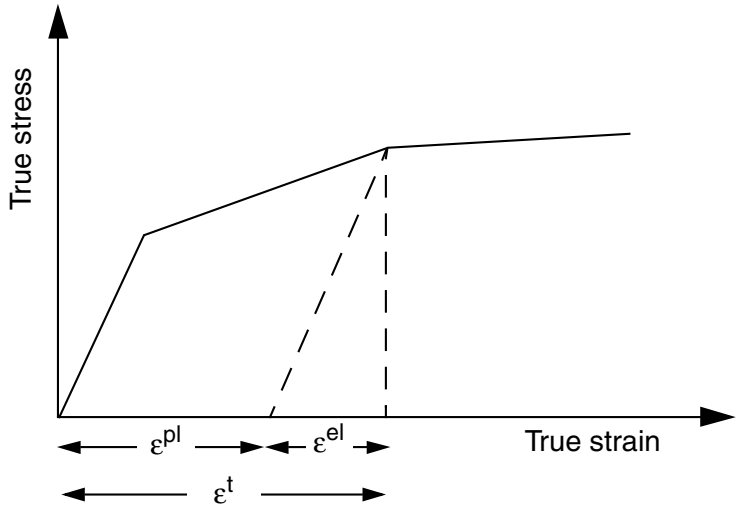


Figure 10–3 Decomposition of the total strain into elastic and plastic components.

This relationship is written

$$\varepsilon^{pl} = \varepsilon^{t} - \varepsilon^{el} = \varepsilon^{t} - \sigma/E,$$

where

ε^{pl}	is true plastic strain,
ε^{t}	is true total strain,
ε^{el}	is true elastic strain,
σ	is true stress, and
E	is Young's modulus.

Example of converting material test data to Abaqus input

The nominal stress-strain curve in Figure 10–4 will be used as an example of how to convert the test data defining a material's plastic behavior into the appropriate input format for Abaqus. The six points shown on the nominal stress-strain curve will be used as the data for the ∗PLASTIC option.

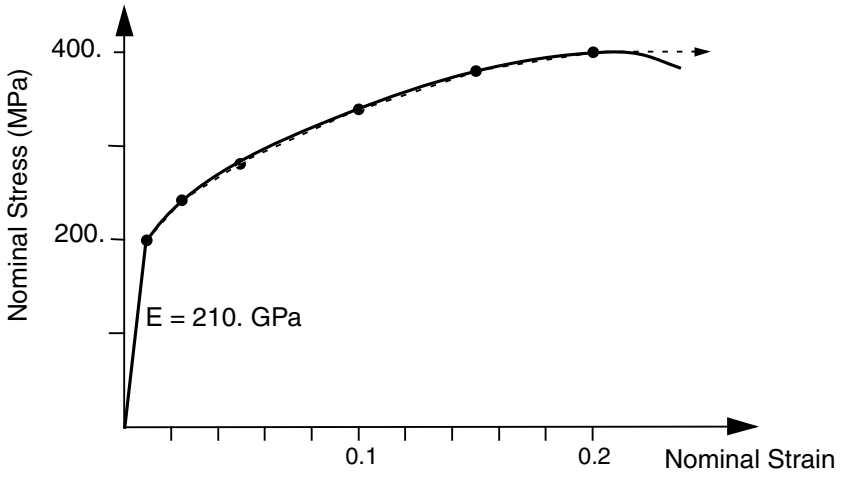


Figure 10–4 Elastic-plastic material behavior.

The first step is to use the equations relating the true stress to the nominal stress and strain and the true strain to the nominal strain (shown earlier) to convert the nominal stress and nominal strain to true stress and true strain. Once these values are known, the equation relating the plastic strain to the total and elastic strains (shown earlier) can be used to determine the plastic strains associated with each yield stress value. The converted data are shown in Table 10–1.

Table 10–1 Stress and strain conversions.

Nominal Stress (Pa)	Nominal Strain	True Stress (Pa)	True Strain	Plastic Strain
200E6	0.00095	200.2E6	0.00095	0.0
240E6	0.025	246E6	0.0247	0.0235
280E6	0.050	294E6	0.0488	0.0474
340E6	0.100	374E6	0.0953	0.0935
380E6	0.150	437E6	0.1398	0.1377
400E6	0.200	480E6	0.1823	0.1800

While there are few differences between the nominal and true values at small strains, there are very significant differences at larger strain values; therefore, it is extremely important to provide the proper stress-strain data to Abaqus if the strains in the simulation will be large.

Data regularization in Abaqus/Explicit

When performing an analysis, Abaqus/Explicit may not use the material data exactly as defined by the user; for efficiency, all material data that are defined in tabular form are automatically *regularized*. Material data can be functions of temperature, external fields, and internal state variables, such as plastic strain. For each material point calculation, the state of the material must be determined by interpolation, and, for efficiency, Abaqus/Explicit fits the user-defined curves with curves composed of equally spaced points. These regularized material curves are the material data used during the analysis. It is important to understand the differences that might exist between the regularized material curves used in the analysis and the curves that you specified.

To illustrate the implications of using regularized material data, consider the following two cases. Figure 10–5 shows a case in which the user has defined data that are not regular.

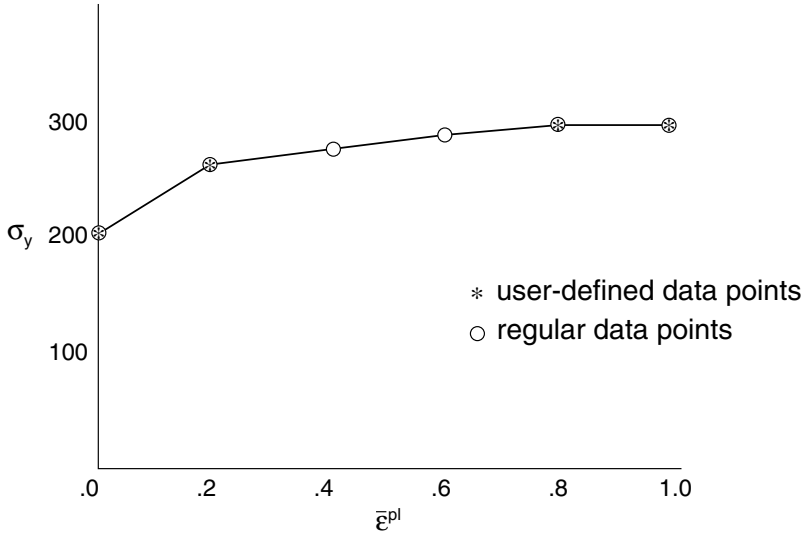


Figure 10–5 Example of user data that can be regularized exactly.

In this example Abaqus/Explicit generates the six regular data points shown, and the user's data are reproduced exactly. Figure 10–6 shows a case where the user has defined data that are difficult to regularize exactly. In this example it is assumed that Abaqus/Explicit has regularized the data by dividing the range into 10 intervals that do not reproduce the user's data points exactly.

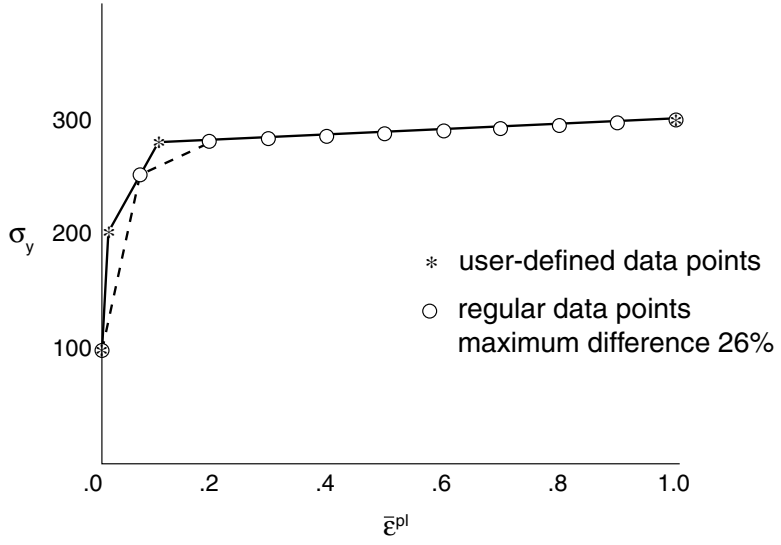


Figure 10–6 Example of user data that are difficult to regularize.

Abaqus/Explicit attempts to use enough intervals such that the maximum error between the regularized data and the user-defined data is less than 3%; however, you can change this error tolerance. If more than 200 intervals are required to obtain an acceptable regularized curve, the analysis stops during the data checking with an error message. In general, the regularization is more difficult if the smallest interval defined by the user is small compared to the range of the independent variable. In Figure 10–6 the data point for a strain of 1.0 makes the range of strain values large compared to the small intervals defined at low strain levels. Removing this last data point enables the data to be regularized much more easily.

Interpolation between data points

Abaqus interpolates linearly between the data points provided (or, in Abaqus/Explicit, regularized data) to obtain the material's response and assumes that the response is constant outside the range defined by the input data, as shown in Figure 10–7. Thus, the stress in this material will never exceed 480 MPa; when the stress in the material reaches 480 MPa, the material will deform continuously until the stress is reduced below this value.

Material calibration in Abaqus/CAE

Abaqus/CAE allows you to calibrate a material model from test data. With this capability, you can import material test data into Abaqus/CAE, process the data, and derive elastic and plastic isotropic material behaviors from the data. This feature is discussed further in "Creating material calibrations," Section 12.17 of the Abaqus/CAE User's Manual.

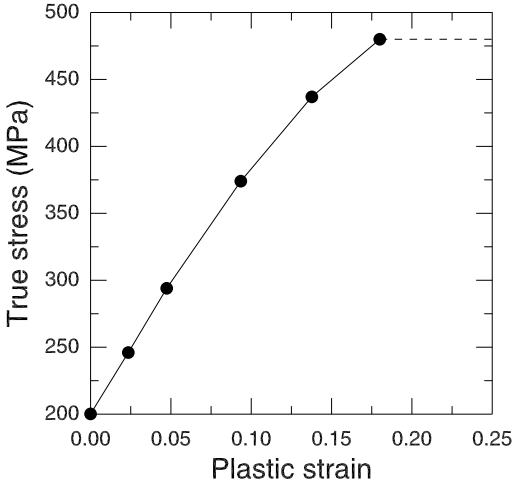


Figure 10–7 Material curve used by Abaqus.

10.3 Selecting elements for elastic-plastic problems

The incompressible nature of plastic deformation in metals places limitations on the types of elements that can be used for an elastic-plastic simulation. The limitations arise because modeling incompressible material behavior adds kinematic constraints to an element; in this case the limitations constrain the volume at the element's integration points to remain constant. In certain classes of elements the addition of these incompressibility constraints makes the element overconstrained. When these elements cannot resolve all of these constraints, they suffer from *volumetric locking*, which causes their response to be too stiff. Volumetric locking is indicated by a rapid variation of hydrostatic pressure stress from element to element or integration point to integration point.

The fully integrated, second-order, solid elements available in Abaqus/Standard are very susceptible to volumetric locking when modeling incompressible material behavior and, therefore, should not be used in elastic-plastic simulations. The fully integrated, first-order, solid elements in Abaqus/Standard do not suffer from volumetric locking because Abaqus actually uses a constant volume strain in these elements. Thus, they can be used safely in plasticity problems.

Reduced-integration solid elements have fewer integration points at which the incompressibility constraints must be satisfied. Therefore, they are not overconstrained and can be used for most elastic-plastic simulations. The second-order reduced-integration elements in Abaqus/Standard should be used with caution if the strains exceed 20–40% because at this magnitude they can suffer from volumetric locking. This effect can be reduced with mesh refinement.

If you have to use fully integrated, second-order elements in Abaqus/Standard, use the hybrid versions, which are designed to model incompressible behavior; however, the additional degrees of freedom in these elements will make the analysis more computationally expensive.

A family of modified second-order triangular and tetrahedral elements is available that provides improved performance over the first-order triangular and tetrahedral elements and that avoids some of the problems that exist for conventional second-order triangular and tetrahedral elements. In particular, these elements exhibit minimal shear and volumetric locking. These elements are available in addition to fully integrated and hybrid elements in Abaqus/Standard; they are the only second-order continuum (solid) elements available in Abaqus/Explicit.

10.4 Example: connecting lug with plasticity

You have been asked to investigate what happens if the steel connecting lug from Chapter 4, "Using Continuum Elements," is subjected to an extreme load (60 kN) caused by an accident. The results from the linear analysis indicate that the lug will yield. You need to determine the extent of the plastic deformation in the lug and the magnitude of the plastic strains so that you can assess whether or not the lug will fail. You do not need to consider inertial effects in this analysis; thus, you will use Abaqus/Standard to examine the static response of the lug.

The only inelastic material data available for the steel are its yield stress (380 MPa) and its strain at failure (0.15). You decide to assume that the steel is perfectly plastic: the material does not harden, and the stress can never exceed 380 MPa (see Figure 10–8).

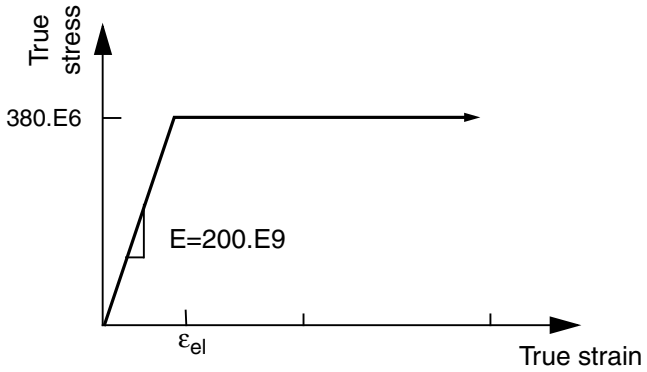


Figure 10–8 Stress-strain behavior for the steel.

In reality some hardening will probably occur, but this assumption is conservative; if the material hardens, the plastic strains will be less than those predicted by the simulation.

The steps that follow assume that you have access to the full input file for this example. This input file, **`lug_plas.inp`**, is provided in "Connecting lug with plasticity," Section A.8, in the online HTML

version of this manual. Instructions on how to fetch and run the script are given in Appendix A, "Example Files."

If you wish to create the entire model using Abaqus/CAE, please refer to "Example: connecting lug with plasticity," Section 10.4 of Getting Started with Abaqus: Interactive Edition.

10.4.1 Modifications to the input file—the model data

In this example, the material definition specifies the post-yield behavior of the material using the *PLASTIC option. The Young's modulus for the material is 200 GPa, and the initial yield stress at zero plastic strain is 380 MPa. Since you are modeling the steel as perfectly plastic, no other yield stresses are given on the *PLASTIC option.

The complete material definition is:

```
*MATERIAL, NAME=STEEL
*ELASTIC
200.E9, 0.3
*PLASTIC
380.E6,0.0
```

All other option blocks in the model definition portion of the input file remain unchanged.

10.4.2 Modifications to the input file—the history data

This analysis requires a general, nonlinear simulation because of the nonlinear material behavior in the model. Therefore, the PERTURBATION parameter must be removed from the *STEP option. The total step time in the *STATIC procedure option block has been set to 1.0, and the initial increment size is 20% of the total step time. This simulation is a static analysis of the lug under the extreme loads; you do not know in advance how many increments this simulation may require. The default maximum of 100 increments, however, is reasonably large and should be sufficient for this analysis. Also, we assume that the effects of geometric nonlinearity will not be important in this simulation, so the NLGEOM parameter is omitted from the *STEP option. This portion of the input file appears as follows.

```
*STEP
*STATIC
0.2, 1.0
```

Loading

The load applied in this simulation is twice what was applied in the linear elastic simulation of the lug (60 kN vs. 30 kN). Therefore, this model doubles the magnitude of the pressures applied to the lug. The modified *DLOAD option block looks like:

```
*DLOAD
PRESS, P6, 1.E+08
```

Output requests

You will use Abaqus/Viewer to review all of the results from this simulation, so all printed output requests have been deleted. The resulting output request option in your input file appears below:

```
*OUTPUT, FIELD, FREQUENCY=1, VARIABLE=PRESELECT
```

You will need to save some history data in the output database file to use with the *X–Y* plotting capability in Abaqus/Viewer. The displacements for node set **HOLEBOT**, which should already exist, are stored using the following option:

```
*OUTPUT, HISTORY, FREQUENCY=1
*NODE OUTPUT, NSET=HOLEBOT
U,
```

You also want detailed results for one of the elements along the built-in end of the lug (see Figure 10–9).

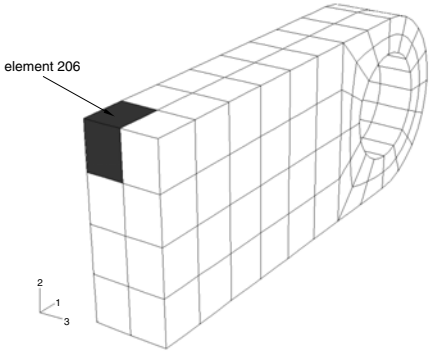


Figure 10–9 Element 206.

This is element 206 in the mesh generated by the commands found in "Connecting lug with plasticity," Section A.8; the element in this location may have a different number in your model. This element is chosen because it is the element for which the stresses are most likely to be largest in magnitude. In the input file for this example, an element set has been created that contains the element, and saves the stresses (S), stress invariants (SINV), plastic strains (PE), and strains (E) for that element set in the output database file. The necessary option blocks are shown below:

```
*ELEMENT OUTPUT, ELSET=EL206
S, SINV, PE, E
*ELSET, ELSET=EL206
206,
```

Note: The *ELSET option used to define the element set appears after the output requests so as to not break up the block of suboptions associated with the *OUTPUT option.

10.4.3 Running the analysis

Save these changes to your model, and run the analysis with the following command:

```
abaqus job=lug_plas
```

Status file

Monitor the simulation while it is running by looking at the status file, **lug_plas.sta**. When Abaqus has finished the simulation, your status file will contain information similar to the following:

```
SUMMARY OF JOB INFORMATION:
STEP   INC ATT SEVERE EQUIL TOTAL   TOTAL       STEP        INC OF        DOF    IF
               DISCON ITERS ITERS   TIME/     TIME/LPF     TIME/LPF    MONITOR RIKS
               ITERS                FREQ
  1     1   1     0     1     1   0.200       0.200       0.2000
  1     2   1     0     1     1   0.400       0.400       0.2000
  1     3   1     0     3     3   0.700       0.700       0.3000
  1     4   1U    0     4     4   0.700       0.700       0.3000
  1     4   2     0     2     2   0.775       0.775       0.07500
  1     5   1     0     4     4   0.887       0.887       0.1125
  1     6   1U    0     4     4   0.887       0.887       0.1125
  1     6   2     0     3     3   0.916       0.916       0.02813
  1     7   1U    0     5     5   0.916       0.916       0.04219
  1     7   2     0     2     2   0.926       0.926       0.01055
  1     8   1     0     4     4   0.942       0.942       0.01582
  1     9   1U    0     3     3   0.942       0.942       0.02373
  1     9   2     0     5     5   0.948       0.948       0.005933
  1    10   1U    0     4     4   0.948       0.948       0.005933
  1    10   2     0     4     4   0.949       0.949       0.001483
  1    11   1     0     4     4   0.951       0.951       0.001483
  1    12   1U    0     3     3   0.951       0.951       0.002225
  1    12   2     0     3     3   0.951       0.951       0.0005562
  1    13   1     0     4     4   0.952       0.952       0.0008343
  1    14   1U    0     2     2   0.952       0.952       0.001251
  1    14   2     0     4     4   0.953       0.953       0.0003129
  1    15   1U    0     2     2   0.953       0.953       0.0004693
  1    15   2     0     3     3   0.953       0.953       0.0001173
  1    16   1U    0     3     3   0.953       0.953       0.0001760
  1    16   2     0     3     3   0.953       0.953       4.399e-005
  1    17   1     0     3     3   0.953       0.953       6.599e-005
  1    18   1U    0     2     2   0.953       0.953       9.899e-005
  1    18   2     0     2     2   0.953       0.953       2.475e-005
  1    19   1     0     3     3   0.953       0.953       3.712e-005
  1    20   1U    0     1     1   0.953       0.953       5.568e-005
  1    20   2     0     3     3   0.953       0.953       1.392e-005
  1    21   1     0     3     3   0.953       0.953       2.088e-005
  1    22   1U    0     1     1   0.953       0.953       3.132e-005
  1    22   2     0     2     2   0.953       0.953       1.000e-005
  1    23   1     0     3     3   0.953       0.953       1.500e-005
  1    24   1U    0     1     1   0.953       0.953       2.250e-005

THE ANALYSIS HAS NOT BEEN COMPLETED
```

Abaqus was able to apply only 95% of the prescribed load to the model and still obtain a converged solution. The status file shows that Abaqus reduced the size of the time increment, which is shown in the last (right-hand) column, many times during the simulation and stopped the analysis in the

24th increment. You will have to look at the information in the message file to understand why Abaqus terminated the simulation early.

Message file

The message file, **lug_plas.msg**, contains detailed information about the simulation's progress (see "Results," Section 8.4.3, for more information about the format of the message file).

Look at the information for the first increment in the analysis (it is also shown below); you will discover that the model's initial behavior is linear, and the displacement correction is ignored because the residual force is essentially zero. The model's behavior was also linear in the second increment.

```
   INCREMENT     1 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  0.200

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1

 AVERAGE FORCE                        128.       TIME AVG. FORCE          128.
 LARGEST RESIDUAL FORCE            -6.684E-10    AT NODE      10605   DOF  2
 LARGEST INCREMENT OF DISP.        -1.685E-04    AT NODE        815   DOF  2
 LARGEST CORRECTION TO DISP.       -1.685E-06    AT NODE        815   DOF  2
          THE FORCE     EQUILIBRIUM RESPONSE WAS LINEAR IN THIS INCREMENT

ITERATION SUMMARY FOR THE INCREMENT:   1 TOTAL ITERATION, OF WHICH
   0 ARE SEVERE DISCONTINUITY ITERATIONS AND  1 ARE EQUILIBRIUM ITERATIONS.
```

Abaqus requires several iterations to obtain a converged solution in the third increment, which indicates that nonlinear behavior occurred in the model during this increment. The only nonlinearity in the model is the plastic material behavior, so the steel must have started to yield somewhere in the lug at this applied load magnitude. The summaries of the iterations for the third increment are shown below.

```
   INCREMENT     3 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  0.300

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1


 AVERAGE FORCE                        794.       TIME AVG. FORCE          459.
 LARGEST RESIDUAL FORCE               831.       AT NODE      13057   DOF  1
 LARGEST INCREMENT OF DISP.        -2.573E-04    AT NODE      20815   DOF  2
 LARGEST CORRECTION TO DISP.       -4.658E-06    AT NODE      10817   DOF  2
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     2


 AVERAGE FORCE                        797.       TIME AVG. FORCE          460.
 LARGEST RESIDUAL FORCE              -23.5       AT NODE      12843   DOF  1
 LARGEST INCREMENT OF DISP.        -2.690E-04    AT NODE      20815   DOF  2
 LARGEST CORRECTION TO DISP.       -1.171E-05    AT NODE       5817   DOF  2
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     3


 AVERAGE FORCE                        801.       TIME AVG. FORCE          461.
 LARGEST RESIDUAL FORCE             1.755E-02    AT NODE      12855   DOF  1
 LARGEST INCREMENT OF DISP.        -2.691E-04    AT NODE        815   DOF  2
 LARGEST CORRECTION TO DISP.       -1.054E-07    AT NODE        817   DOF  2
          THE FORCE     EQUILIBRIUM EQUATIONS HAVE CONVERGED

 ITERATION SUMMARY FOR THE INCREMENT:   3 TOTAL ITERATIONS, OF WHICH
```

```
   0 ARE SEVERE DISCONTINUITY ITERATIONS AND  3 ARE EQUILIBRIUM ITERATIONS.

 TIME INCREMENT COMPLETED  0.300    ,  FRACTION OF STEP COMPLETED  0.700
 STEP TIME COMPLETED       0.700    ,  TOTAL TIME COMPLETED        0.700
```

Abaqus attempts to find a solution in the fourth increment using an increment size of 0.3, which means it is applying 30% of the total load, or 18 MPa, during this increment. After several iterations, Abaqus issues warning messages that the strain increments it calculated exceed the strain at initial yield by 50 times. After a few more iterations Abaqus determines that the solution in this increment is not going to converge; instead, it is diverging. Therefore, Abaqus abandons this attempt at finding a solution, reduces the increment size to 25% of the value used in the first attempt, and tries a second attempt at finding a solution. This reduction in increment size is called a *cut-back*. With the smaller increment size, Abaqus finds a converged solution in just a few iterations. Some of the iteration summaries from the first attempt of the fourth increment are shown below.

```
   INCREMENT     4 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  0.300

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1

 AVERAGE FORCE                      1.196E+03   TIME AVG. FORCE        645.
 LARGEST RESIDUAL FORCE            -4.908E+03   AT NODE      12849   DOF  2
 LARGEST INCREMENT OF DISP.        -5.806E-04   AT NODE      10817   DOF  2
 LARGEST CORRECTION TO DISP.       -3.116E-04   AT NODE      10817   DOF  2
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     2

 AVERAGE FORCE                      1.286E+03   TIME AVG. FORCE        668.
 LARGEST RESIDUAL FORCE             1.168E+04   AT NODE      13045   DOF  1
 LARGEST INCREMENT OF DISP.        -1.484E-03   AT NODE      10817   DOF  2
 LARGEST CORRECTION TO DISP.       -9.038E-04   AT NODE      10817   DOF  2
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     3

 AVERAGE FORCE                      1.486E+03   TIME AVG. FORCE        717.
 LARGEST RESIDUAL FORCE             1.721E+04   AT NODE      13049   DOF  2
 LARGEST INCREMENT OF DISP.        -6.796E-03   AT NODE      10817   DOF  2
 LARGEST CORRECTION TO DISP.       -5.311E-03   AT NODE        817   DOF  2
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

***WARNING: THE STRAIN INCREMENT HAS EXCEEDED FIFTY TIMES THE STRAIN TO CAUSE
            FIRST YIELD AT 120 POINTS

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     4

 AVERAGE FORCE                      2.356E+03   TIME AVG. FORCE        935.
 LARGEST RESIDUAL FORCE            -4.587E+04   AT NODE      12447   DOF  1
 LARGEST INCREMENT OF DISP.        -7.530E-02   AT NODE      10817   DOF  2
 LARGEST CORRECTION TO DISP.       -6.850E-02   AT NODE       5817   DOF  2
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.


***NOTE: THE SOLUTION APPEARS TO BE DIVERGING. CONVERGENCE IS JUDGED UNLIKELY.
```

Now that you have reviewed the early increments of the simulation, move to the end of the message file and review the last increment Abaqus attempted. You will see that Abaqus is using a very small increment size, on the order of 1.0×10^{-5}, in this final increment because of the many cut-backs. The iteration summaries for the last increment are shown below. Abaqus makes two attempts to find a solution in this final increment, but it must cut back the time increment in each

attempt because the strain increments are so large that it does not even try to perform the plasticity calculations. This check on the magnitude of the total strain increment is another example of the many automatic solution controls Abaqus uses to ensure that the solution obtained for your simulation is both accurate and efficient. The automatic solution controls are suitable for almost all simulations. Therefore, you do not have to worry about providing parameters to control the solution algorithm: you only have to be concerned with the input data for your model.

```
   INCREMENT    24 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  2.250E-05

***WARNING: THE STRAIN INCREMENT HAS EXCEEDED FIFTY TIMES THE STRAIN TO CAUSE
            FIRST YIELD AT 152 POINTS

***WARNING: THE STRAIN INCREMENT IS SO LARGE THAT THE PROGRAM WILL NOT ATTEMPT
            THE PLASTICITY CALCULATION AT 16 POINTS

***NOTE: MATERIAL CALCULATIONS FAILED TO CONVERGE OR WERE NOT ATTEMPTED
         AT ONE OR MORE POINTS. CONVERGENCE IS JUDGED UNLIKELY.


   INCREMENT    24 STARTS. ATTEMPT NUMBER  2, TIME INCREMENT  1.000E-05

***WARNING: THE STRAIN INCREMENT HAS EXCEEDED FIFTY TIMES THE STRAIN TO CAUSE
            FIRST YIELD AT 120 POINTS

***WARNING: THE STRAIN INCREMENT HAS EXCEEDED FIFTY TIMES THE STRAIN TO CAUSE
            FIRST YIELD AT 132 POINTS

               CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1

 AVERAGE FORCE                      1.751E+03   TIME AVG. FORCE       1.352E+03.
 LARGEST RESIDUAL FORCE              -44.9      AT NODE      11841   DOF  2
 LARGEST INCREMENT OF DISP.         -0.153      AT NODE      15817   DOF  2
 LARGEST CORRECTION TO DISP.       -4.662E-02   AT NODE      10817   DOF  2
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

***WARNING: THE STRAIN INCREMENT HAS EXCEEDED FIFTY TIMES THE STRAIN TO CAUSE
            FIRST YIELD AT 136 POINTS

***WARNING: THE STRAIN INCREMENT IS SO LARGE THAT THE PROGRAM WILL NOT ATTEMPT
            THE PLASTICITY CALCULATION AT 4 POINTS

***NOTE: MATERIAL CALCULATIONS FAILED TO CONVERGE OR WERE NOT ATTEMPTED
              AT ONE OR MORE POINTS. CONVERGENCE IS JUDGED UNLIKELY.


***ERROR: TIME INCREMENT REQUIRED IS LESS THAN THE MINIMUM SPECIFIED
```

If you look at the summary at the end of the message file, you will find that Abaqus issued many warning messages during the analysis. Reviewing the message file will show that most of these warnings were the result of numerical problems with the plasticity calculations. You know that Abaqus terminated the analysis early because these numerical problems forced it to cut back the time increment until it was below the minimum allowable time increment.

In the third column of the status file you will see the number of attempts Abaqus made to solve an increment. In the sixth column the number of iterations needed for the last attempt at an increment is printed. Now, you should look at the results in Abaqus/Viewer to understand what caused this excessive plasticity.

10.4.4 Postprocessing the results

Look at the results in Abaqus/Viewer to understand what caused the excessive plasticity. Run Abaqus/Viewer by entering the following command at the operating system prompt:

```
abaqus viewer odb=lug_plas
```

Plotting the deformed model shape

Create a plot of the model's deformed shape, and check that this shape is realistic.

The default view is isometric. You can set the view shown in Figure 10–10 by using the options in the **View** menu or the tools in the **View Manipulation** toolbar; in this figure perspective is also turned off.

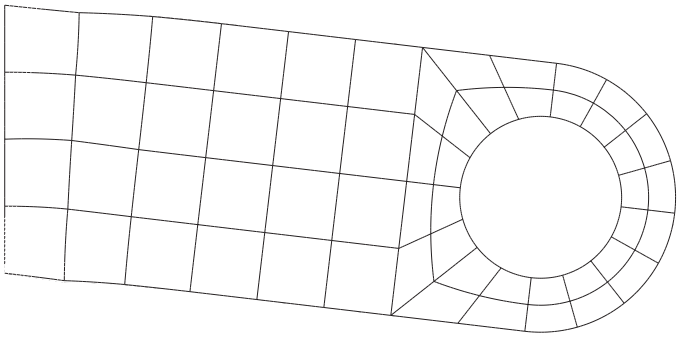


Figure 10–10 Deformed model shape using results for the simulation without hardening.

The displacements and, particularly, the rotations of the lug shown in the plot are large but do not seem large enough to have caused all of the numerical problems seen in the simulation. Look closely at the information in the plot's title for an explanation. The deformation scale factor used in this plot is 0.02; i.e., the displacements are scaled to 2% of their actual values. (Your deformation scale factor may be different.)

Abaqus/Viewer always scales the displacements in a geometrically linear simulation such that the deformed shape of the model fits into the viewport. (This is in contrast to a geometrically nonlinear simulation, where Abaqus/Viewer does not scale the displacements and, instead, adjusts the view by zooming in or out to fit the deformed shape in the plot.) To plot the actual displacements,

set the deformation scale factor to 1.0. This will produce a plot of the model in which the lug has deformed until it is almost parallel to the vertical (global *Y*) axis.

The applied load of 60 kN exceeds the limit load of the lug, and the lug collapses when the material yields at all the integration points through its thickness. The lug then has no stiffness to resist further deformation because of the perfectly plastic post-yield behavior of the steel. This is consistent with the pattern observed earlier concerning the locations of the large strain increments and maximum displacement corrections.

10.4.5 Adding hardening to the material model

The connecting lug simulation with perfectly plastic material behavior predicts that the lug will suffer catastrophic failure caused by the collapse of the structure. We have already mentioned that the steel would probably exhibit some hardening after it has yielded. You suspect that including hardening behavior would allow the lug to withstand this 60 kN load because of the additional stiffness it would provide. Therefore, you decide to add some hardening to the steel's material property definition. Assume that the yield stress increases to 580 MPa at a plastic strain of 0.35, which represents typical hardening for this class of steel. The stress-strain curve for the modified material model is shown in Figure 10–11.

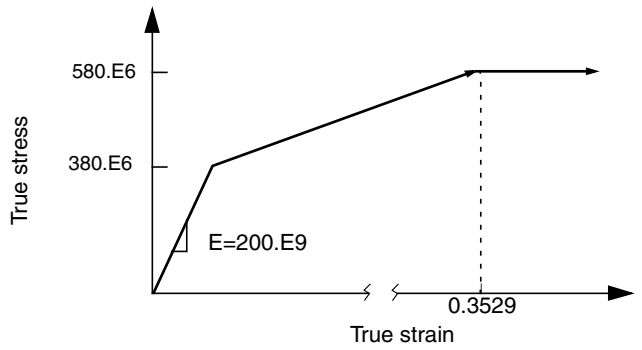


Figure 10–11 Modified stress-strain behavior of the steel.

Modify your *PLASTIC option block as follows so that it includes the hardening data:

```
*PLASTIC
380.E6, 0.00
580.E6, 0.35
```

10.4.6 Running the analysis with plastic hardening

Save the model with plastic hardening to a new input file named **lug_plas_hard.inp**, and run the analysis using the command

```
abaqus job=lug_plas_hard
```

Status file

The summary of the analysis in the status file, which is shown below, shows that Abaqus found a converged solution when the full 60 kN load was applied. The hardening data added enough stiffness to the lug to prevent it from collapsing under the 60 kN load.

```
SUMMARY OF JOB INFORMATION:
STEP   INC ATT SEVERE EQUIL TOTAL  TOTAL      STEP       INC OF       DOF    IF
               DISCON ITERS ITERS  TIME/    TIME/LPF    TIME/LPF    MONITOR RIKS
               ITERS               FREQ
  1     1   1     0     1     1  0.200       0.200       0.2000
  1     2   1     0     1     1  0.400       0.400       0.2000
  1     3   1     0     3     3  0.700       0.700       0.3000
  1     4   1     0     7     7  1.00        1.00        0.3000
```

In this simulation there is very little information of interest in the message file. There are no warnings issued during the analysis, so you can proceed directly to postprocessing the results with Abaqus/Viewer.

10.4.7 Postprocessing the results

Start Abaqus/Viewer, and use the following command to review the results of the second analysis:

```
abaqus viewer odb=lug_plas_hard
```

Deformed model shape and peak displacements

Plot the deformed model shape with these new results, and change the deformation scale factor to 2 to obtain a plot similar to Figure 10–12. The displayed deformations are double the actual deformations.

Contour plot of Mises stress

Contour the Mises stress in the model. Create a filled contour plot using ten contour intervals on the actual deformed shape of the lug (i.e., set the deformation scale factor to 1.0) with the plot title and state blocks suppressed. Use the view manipulation tools to position and size the model to obtain a plot similar to that shown in Figure 10–13.

Do the values listed in the contour legend surprise you? The maximum stress is greater than 580 MPa, which should not be possible since the material was assumed to be perfectly plastic at this stress magnitude. This misleading result occurs because of the algorithm that Abaqus/Viewer uses to create contour plots for element variables, such as stress. The contouring algorithm requires data at the nodes; however, Abaqus/Standard calculates element variables at the integration points. Abaqus/Viewer calculates nodal values of element variables by extrapolating the data from

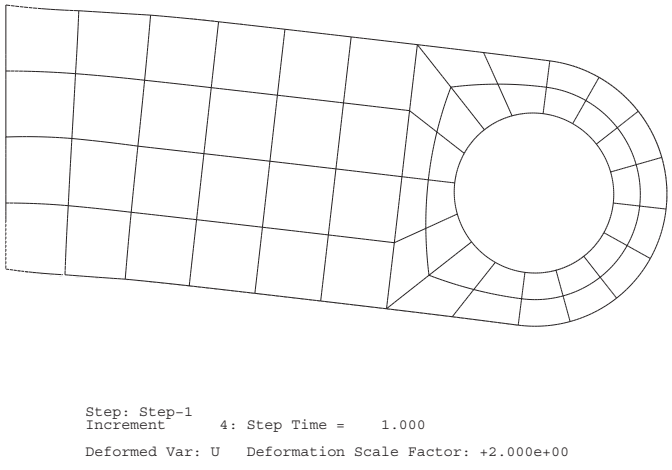


Figure 10–12 Deformed model shape for the simulation with plastic hardening.

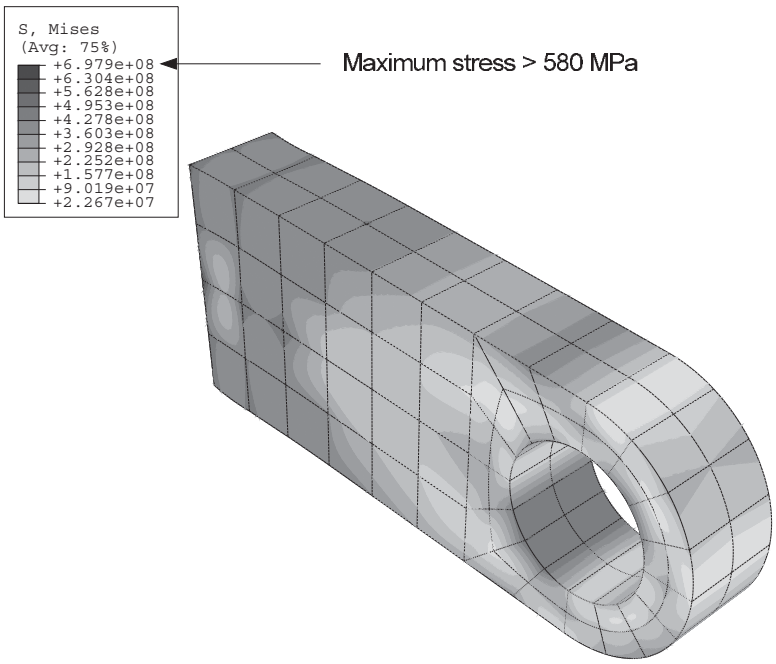


Figure 10–13 Contour of Mises stress.

the integration points to the nodes. The extrapolation order depends on the element type; for

second-order, reduced-integration elements Abaqus/Viewer uses linear extrapolation to calculate the nodal values of element variables. To display a contour plot of Mises stress, Abaqus/Viewer extrapolates the stress components from the integration points to the nodal locations within each element and calculates the Mises stress. If the differences in Mises stress values fall within the specified averaging threshold, nodal averaged Mises stresses are calculated from each surrounding element's invariant stress value. Invariant values exceeding the elastic limit can be produced by the extrapolation process.

Try plotting contours of each component of the stress tensor (variables S11, S22, S33, S12, S23, and S13). You will see that there are significant variations in these stresses across the elements at the built-in end. This causes the extrapolated nodal stresses to be higher than the values at the integration points. The Mises stress calculated from these values will, therefore, also be higher.

Note: Element type C3D10I does not suffer from this extrapolation problem. The integration points of this element type are located at the nodes, resulting in improved surface stress visualization.

The Mises stress at an integration point can never exceed the current yield stress of the element's material; however, the extrapolated nodal values reported in a contour plot may do so. In addition, the individual stress components may have magnitudes that exceed the value of the current yield stress; only the Mises stress is required to have a magnitude less than or equal to the value of the current yield stress.

You can use the query tools in the Visualization module to check the Mises stress at the integration points.

To query the Mises stress:

1. From the main menu bar, select **Tools→Query**; or use the tool in the **Query** toolbar.
   The **Query** dialog box appears.
2. In the **Visualization Module Queries** field, select **Probe values**.
   The **Probe Values** dialog box appears.
3. Make sure that **Elements** and the output position **Integration Pt** are selected.
4. Use the cursor to select elements near the constrained end of the lug.
   Abaqus/Viewer reports the element ID and type by default and the value of the Mises stress at each integration point starting with the first integration point. The Mises stress values at the integration points are all lower than the values reported in the contour legend and also below the yield stress of 580 MPa. You can click mouse button 1 to store probed values.
5. Click **Cancel** when you have finished probing the results.

The fact that the extrapolated values are so different from the integration point values indicates that there is a rapid variation of stress across the elements and that the mesh is too coarse for accurate stress calculations. This extrapolation error will be less significant if the mesh is refined but will

always be present to some extent. Therefore, always use nodal values of element variables with caution.

Contour plot of equivalent plastic strain

The equivalent plastic strain in a material (PEEQ) is a scalar variable that is used to represent the material's inelastic deformation. If this variable is greater than zero, the material has yielded. Those parts of the lug that have yielded can be identified in a contour plot of PEEQ by selecting **Primary** from the list of variable types on the left side of the **Field Output** toolbar and selecting **PEEQ** from the list of output variables. Any areas in the model plotted in a dark color in Abaqus/Viewer still have elastic material behavior (see Figure 10–14).

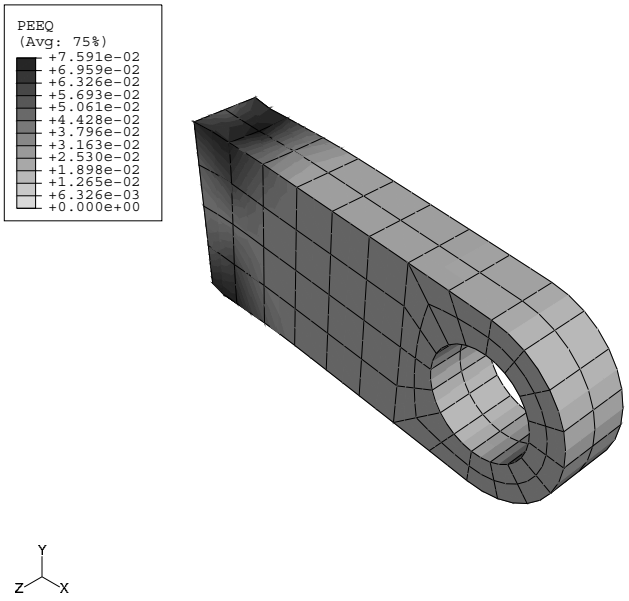


Figure 10–14 Contour of equivalent plastic strain (PEEQ).

It is clear from the plot that there is gross yielding in the lug where it is attached to its parent structure. The maximum plastic strain reported in the contour legend is about 10%. However, this value may contain errors from the extrapolation process. Use the query tool to check the integration point values of PEEQ in the elements with the largest plastic strains. You will find that the largest equivalent plastic strains in the model are about 0.067 at the integration points. This does not necessarily indicate a large extrapolation error since there are strain gradients present in the vicinity of the peak plastic deformation.

Creating a variable-variable (stress-strain) plot

The *X–Y* plotting capability in Abaqus/Viewer was introduced earlier in this manual. In this section you will learn how to create *X–Y* plots showing the variation of one variable as a function of another. You will use the stress and strain data saved to the output database (**`.odb`**) file to create a stress-strain plot for one of the integration points in an element adjacent to the constrained end of the lug.

In the model used in this discussion, the stress-strain data were saved for element 206. You may have specified a different element number in your model; if you did, use that element number in place of 206 in the input examples that follow. Also, use data from an integration point that is closest to the top surface of the lug but not adjacent to the constrained nodes. Thus, you will need to identify the element's label as well as its nodal connectivity to determine which integration point to use.

To determine the integration point number:

1. In the **Display Group** toolbar, select the **Create Display Group** tool; in the **Create Display Group** dialog box, select **Elements** as the item and **Element sets** as the method. From the list of available element sets, select **EL206** and toggle on **Highlight items in viewport** to confirm its selection. Click **Replace**.
2. Plot the undeformed shape of this element with the node labels made visible. Click the auto-fit tool to obtain a plot similar to Figure 10–15.
3. Use the **Query** tool to obtain the nodal connectivity for this corner element (select **Element**, and click the element in the viewport). The nodal connectivity will be printed to the message area; you are interested in only the first four nodes.
4. Compare the nodal connectivity list with the undeformed model shape plot to determine which is the 1–2–3–4 face on your C3D20R element, as defined in "Three-dimensional solid element library," Section 28.1.4 of the Abaqus Analysis User's Manual. For example, in Figure 10–15 the 2841, 3241, 3245, 2845 face corresponds to the 1–2–3–4 face. After comparing these, you will find we are interested in the point that corresponds to integration point 1.

To create history curves of stress and direct strain along the lug in element 206:

1. In the Results Tree, click mouse button 3 on **History Output** for the output database named **`lug_plas_hard.odb`**. From the menu that appears, select **Filter**.
2. In the filter field, enter **`*MISES*`** to restrict the history output to just the Mises stress.
3. Click mouse button 3 on the MISES stress at element 206, integration point 1. From the menu that appears, select **Save As**. Enter the name **`MISES`** and click **OK**.
4. Filter the history output using **`*E11*`** and save the **E11** strain component at the same integration point. Name the curve **`E11`**.

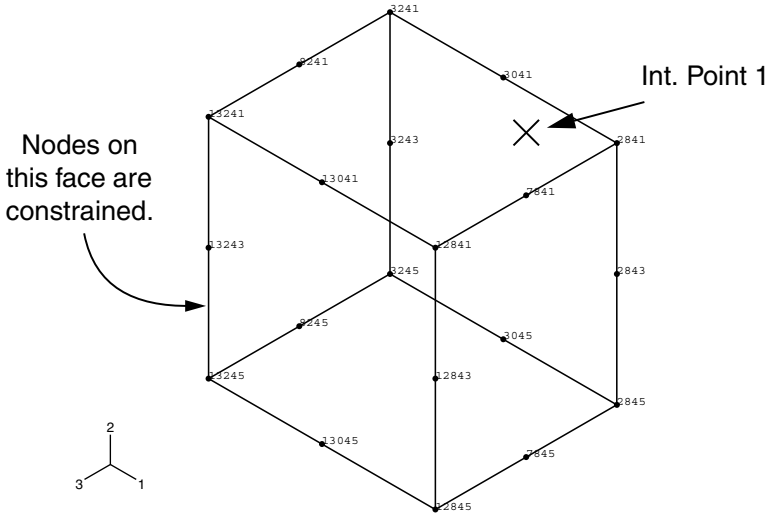


Figure 10–15 Location of integration point 1 in element 206.

The **MISES** stress, rather than the component of the true stress tensor, is used because the plasticity model defines plastic yield in terms of Mises stress. The **E11** strain component is used because it is the largest component of the total strain tensor at this point; using it clearly shows the elastic, as well as the plastic, behavior of the material at this integration point.

The curves appear in the **XYData** container. Each of the curves is a history (variable versus time) plot. You must combine the plots, eliminating the time dependence, to produce the desired stress-strain plot.

To combine history curves to produce a stress-strain plot:

1. In the Results Tree, double-click **XYData**.

   The **Create XY Data** dialog box appears.

2. Select **Operate on XY data**, and click **Continue**.

   The **Operate on XY Data** dialog box appears. Expand the **Name** field to see the full name of each curve.

3. From the **Operators** listed, select **combine(X,X)**.

   `combine( )` appears in the text field at the top of the dialog box.

4. In the **XY Data** field, select the stress and strain curves.

5. Click **Add to Expression**. The expression `combine("E11", "MISES")` appears in the text field. In this expression `"E11"` will determine the *X*-values and `"MISES"` will determine the *Y*-values in the combined plot.
6. Save the combined data object by clicking **Save As** at the bottom of the dialog box.

   The **Save XY Data As** dialog box appears. In the **Name** text field, type `SVE11`; and click **OK** to close the dialog box.
7. To view the combined stress-strain plot, click **Plot Expression** at the bottom of the dialog box.
8. Click **Cancel** to close the dialog box.
9. Click in the prompt area to cancel the current procedure.

   This *X*–*Y* plot would be clearer if the limits on the *X*- and *Y*-axes were changed.

To customize the stress-strain curve:

1. Double-click either axis to open the **Axis Options** dialog box.
2. Set the maximum value of the *X*-axis (E11 strain) to `0.09`, the maximum value of the *Y*-axis (MISES stress) to `500` MPa, and the minimum value to `0.0` MPa.
3. Switch to the **Title** tabbed page, and customize the *X*- and *Y*-axis labels as shown in Figure 10–16.
4. Click **Dismiss** to close the **Axis Options** dialog box.
5. It will also be helpful to display a symbol at each data point of the curve. Open the **Curve Options** dialog box.
6. From the **Curves** field, select the stress-strain curve (`SVE11`).

   The `SVE11` data object is highlighted.
7. Toggle on **Show symbol**. Accept the defaults, and click **Dismiss** at the bottom of the dialog box.

   The stress-strain plot appears with a symbol at each data point of the curve.

You should now have a plot similar to the one shown in Figure 10–16. The stress-strain curve shows that the material behavior was linear elastic for this integration point during the first two increments of the simulation. In this plot it appears that the material remains linear during the third increment of the analysis; however, it does yield during this increment. This illusion is created by the extent of strain shown in the plot. If you limit the maximum strain displayed to 0.01 and set the minimum value to 0.0, the nonlinear material behavior in the third increment can be seen more clearly (see Figure 10–17).

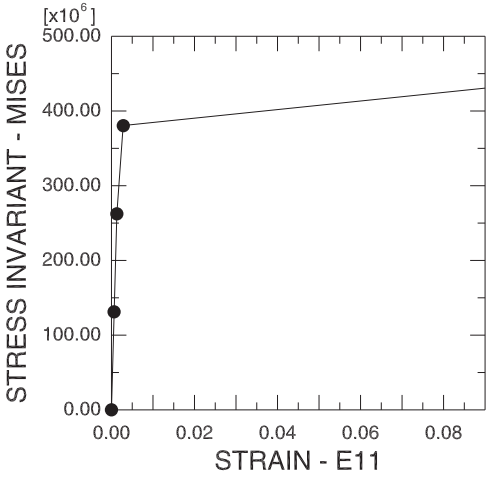


Figure 10–16 Mises stress vs. direct strain (E11) along the lug in the corner element.

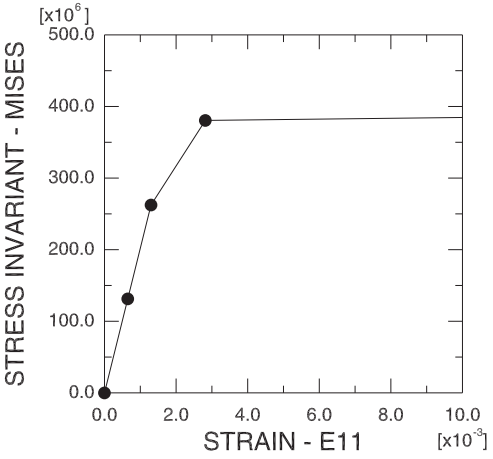


Figure 10–17 Mises stress vs. direct strain (E11) along the lug in the corner element. Maximum strain 0.01.

This stress-strain curve contains another apparent error. It appears that the material yields at 250 MPa, which is well below the initial yield stress. However, this error is caused by the fact that Abaqus/Viewer connects the data points on the curve with straight lines. If you limit the increment size, the additional points on the graph will provide a better display of the material response and show yield occurring at exactly 380 MPa.

The results from this second simulation indicate that the lug will withstand this 60 kN load if the steel hardens after it yields. Taken together, the results of the two simulations demonstrate that it is very important to determine the actual post-yield hardening behavior of the steel. If the steel has very little hardening, the lug may collapse under the 60 kN load. Whereas if it has moderate hardening, the lug will probably withstand the load although there will be extensive plastic yielding in the lug (see Figure 10–14). However, even with plastic hardening, the factor of safety for this loading will probably be very small.

10.5 Example: blast loading on a stiffened plate

The previous example illustrated some of the convergence difficulties that may be encountered when solving problems involving a nonlinear material response using implicit methods. We will now focus on solving a problem involving plasticity using explicit dynamics. As will become evident shortly, convergence difficulties are not an issue in this case since iteration is not required for explicit methods.

In this example you will assess the response of a stiffened square plate subjected to a blast loading in Abaqus/Explicit. The plate is firmly clamped on all four sides and has three equally spaced stiffeners welded to it. The plate is constructed of 25 mm thick steel and is 2 m square. The stiffeners are made from 12.5 mm thick plate and have a depth of 100 mm. Figure 10–18 shows the plate geometry and material properties in more detail. Since the plate thickness is significantly smaller than any other global dimensions, shell elements can be used to model the plate.

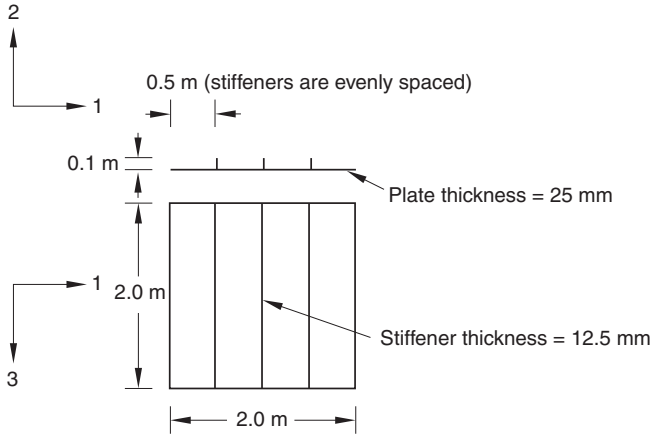
The purpose of this example is to determine the response of the plate and to see how it changes as the sophistication of the material model increases. Initially, we analyze the behavior with the standard elastic-plastic material model. Subsequently, we study the effects of including material damping and rate-dependent material properties.

10.5.1 Coordinate system

This model uses the default rectangular coordinate system with the plate lying in the 1–3 plane. Since the plate thickness is significantly smaller than any other global dimensions, we can use shell elements of type S4R for the model.

10.5.2 Mesh design

The mesh in this model is based on the design shown in Figure 10–19, which is a relatively coarse mesh of 20 × 20 elements in the plate and 2 × 20 elements in each of the stiffeners. This mesh corresponds to the input file shown in “Blast loading on a stiffened plate,” Section A.9. It provides moderate accuracy while keeping the solution time to a minimum. Define the mesh so that the element normals for the plate



Material properties

General properties:

$\rho = 7800 \text{ kg/m}^3$

Elastic properties:

$E = 210 \times 10^9 \text{ Pa}$

$\nu = 0.3$

Plastic properties:

True Stress (Pa)	True Plastic Strain
300×10^6	0.000
350×10^6	0.025
375×10^6	0.100
394×10^6	0.200
400×10^6	0.350

Figure 10–18 Problem description for blast load on a stiffened plate.

all point in the positive 2-direction. Doing so ensures that the stiffeners lie on the SPOS face of the plate, which will be important when defining the element properties and shell offsets later.

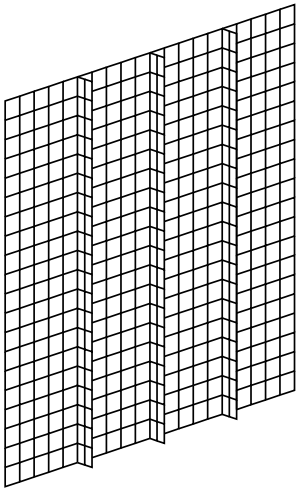


Figure 10–19 Mesh design for the stiffened plate.

10.5.3 Node and element sets

The steps that follow assume that you have access to the full input file for this example. This input file, **blast_base.inp**, is provided in "Example: blast loading on a stiffened plate," Section 10.5, in the online HTML version of this manual. Instructions on how to fetch and run the script are given in Appendix A, "Example Files."

If you wish to create the entire model using Abaqus/CAE, please refer to "Example: blast loading on a stiffened plate," Section 10.5 of Getting Started with Abaqus: Interactive Edition.

Figure 10–20 shows all the sets necessary to apply the element properties, loads, and boundary conditions. This model includes all the nodes on the perimeter of the plate in a node set called **EDGE**. These nodes will have a completely fixed boundary condition. For output purposes, a node set called **NOUT** has been created containing the node at the center of the plate. Plate elements are included in an element set called **PLATE**, and the stiffener elements in an element set called **STIFF**. In addition, the four center elements on the central stiffener are included in an element set called **STIFFMAX**; this element set is for output purposes. These center elements will be subject to the maximum bending stress in the stiffeners.

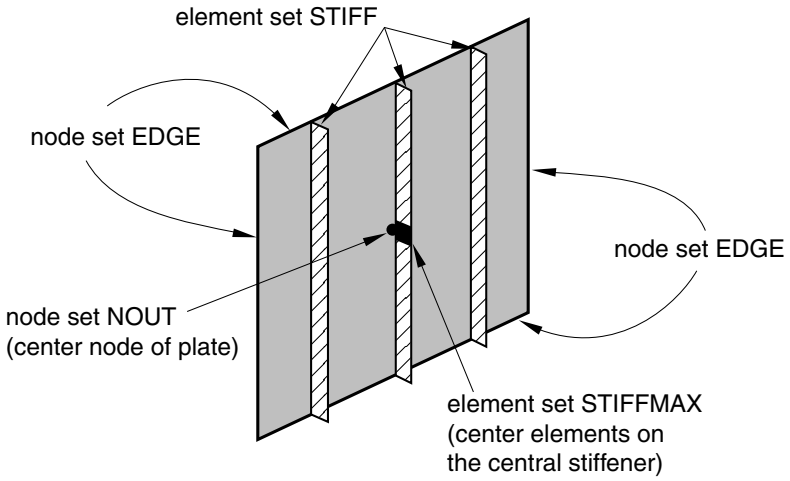


Figure 10–20 Node and element sets.

10.5.4 Reviewing the input file—the model data

We now review the model data for this problem, including the model description, node and element definitions, element properties and shell offsets, material properties, boundary conditions, and amplitude definition for the blast load.

Model description

The *HEADING option is used to include a title and model description in the input file. The heading is useful for future reference purposes and may contain information on model revisions and the evolution of complex models. It can be several lines long, but only the first line will be printed as a title on the output pages. Below is the *HEADING definition used for this analysis.

```
*HEADING
Blast load on a flat plate with stiffeners
S4R elements (20x20 mesh)
Normal stiffeners (20x2)
SI units (kg, m, s, N)
```

Nodal coordinates and element connectivity

The mesh is shown in Figure 10–19 and the sets are shown in Figure 10–20.

Element properties and shell offset

Each element set in the model has the section properties shown below. To insure that each set of elements refers to a material definition, the appropriate MATERIAL parameter has been included on each *SHELL SECTION option:

```
*SHELL SECTION, MATERIAL=STEEL, ELSET=PLATE, OFFSET=SPOS
0.025,
*SHELL SECTION, MATERIAL=STEEL, ELSET=STIFF
0.0125,
```

The material named **STEEL** will be defined in the next section. Setting OFFSET to SPOS offsets the midsurface of the plate one half of the shell thickness away from the nodes. The effect is to make the **PLATE** nodes lie on the SPOS shell face instead of on the shell midsurface. The purpose of the shell offset in this case is to allow the stiffeners to butt up against the plate without overlapping any material with the plate. Figure 10–21 shows the cross-section of the joint between the stiffener and panel using the OFFSET parameter.

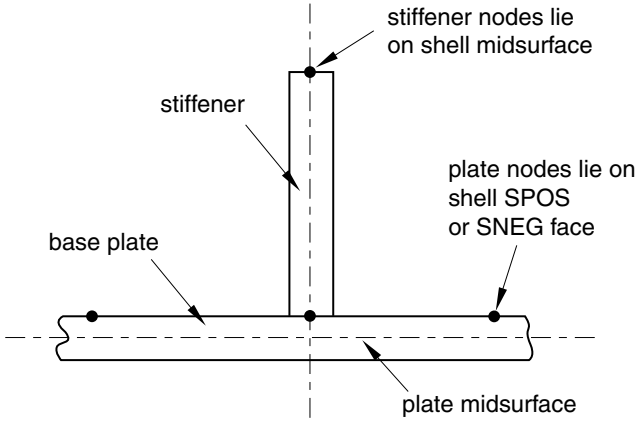


Figure 10–21 Stiffener joint in which the plate's midsurface is offset from its nodes.

If the stiffener and base plate elements are joined at common nodes at their midsurfaces, an area of material overlaps, as shown in Figure 10–22.

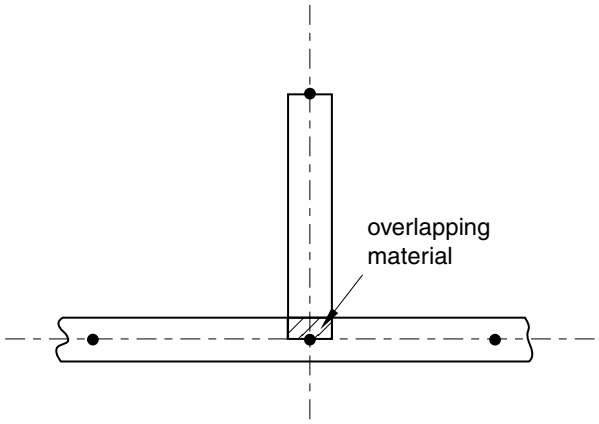


Figure 10–22 Overlapping material if OFFSET were not used.

If the thicknesses of the plate and stiffener is small in comparison to the overall dimensions of the structure, this overlapping material and the extra stiffness it would create has little effect on the analysis results. However, if the stiffener is short in comparison to its width or to the thickness of the base plate, the additional stiffness of the overlapping material could affect the response of the whole structure.

Material properties

Assume that both the plate and stiffeners are made of steel (Young's modulus of 210.0 GPa and Poisson's ratio of 0.3). At this stage we do not know whether there will be any plastic deformation, but we know the value of the yield stress and details of the post-yield behavior for this steel. We will add this information on the *PLASTIC option in the material definition. The initial yield stress is 300 MPa, and the yield stress increases to 400 MPa at a plastic strain of 35%. The plasticity data are shown below, and the plasticity stress-strain curve is shown in Figure 10–23.

```
*MATERIAL, NAME=STEEL
*ELASTIC
210.0E9, .3
*PLASTIC
300.0E6, 0.000
350.0E6, 0.025
375.0E6, 0.100
394.0E6, 0.200
400.0E6, 0.350
*DENSITY
7800.0,
```

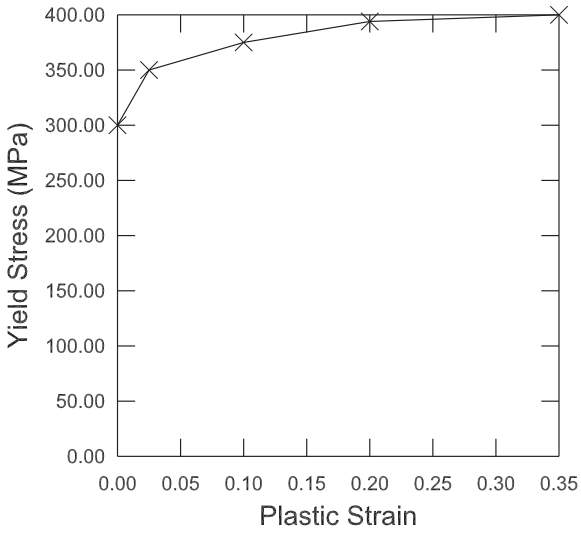


Figure 10–23 Yield stress versus plastic strain.

During the analysis Abaqus calculates values of yield stress from the current values of plastic strain. As discussed earlier, the process of lookup and interpolation is most efficient when the data are regular—when the stress-strain data are at equally spaced values of plastic strain. To avoid having the user input regular data, Abaqus/Explicit automatically regularizes the data. In this case the data are regularized by Abaqus/Explicit by expanding to 15 equally spaced points with increments of 0.025.

To illustrate the error message that is produced when Abaqus/Explicit cannot regularize the material data, try setting the regularization tolerance, RTOL, to 0.001 and include one additional data pair, as shown below:

```
*MATERIAL, NAME=STEEL, RTOL=0.001
*ELASTIC
210.0E9,  .3
*PLASTIC
300.0E6,  0.0
349.0E6,  0.001 <-- additional data pair
350.0E6,  0.025
375.0E6,  0.10
394.0E6,  0.20
400.0E6,  0.35
```

The combination of the low tolerance value (RTOL=0.001) and the small interval in the user-defined data leads to difficulty in regularizing this material definition. The following error message is produced in the status (**.sta**) file:

```
 ***ERROR: Failed to regularize material data. Please check
 your input data to see if they meet both criteria as
 explained in the "MATERIAL DEFINITION" section of the
Abaqus Analysis User's Manual. In general, regularization is
 more difficult if the smallest interval defined by the user
 is small compared to the range of the independent variable.
```

Before continuing, set the regularization tolerance back to the default value (0.03) and remove the additional pair of data points.

Boundary conditions

The edges of the plate are fully constrained using the node set **EDGE** defined previously.

```
*BOUNDARY
EDGE, ENCASTRE
```

Alternatively, you could specify the degrees of freedom by number.

```
*BOUNDARY
EDGE, 1, 6
```

Amplitude definition for blast load

Since the plate will be subjected to a load that varies with time, you must define an appropriate amplitude curve to describe the variation. The amplitude curve shown in Figure 10–24 can be defined as follows:

```
*AMPLITUDE, NAME=BLAST
0.0, 0.0, 1.0E-3, 7.0E5, 10E-3, 7.0E5, 20E-3, 0.0
50E-3, 0.0
```

The pressure increases rapidly from zero at the start of the analysis to its maximum of 7.0×10^5 Pa in 1 ms, at which point it remains constant for 9 ms before dropping back to zero in another 10 ms. It then remains at zero for the remainder of the analysis.

10.5.5 Reviewing the input file—the history data

The history data begin with the *STEP option, which is followed immediately by a title for the step. After the title, this step is defined as a *DYNAMIC, EXPLICIT procedure with a time period of 50 ms.

```
*STEP
Apply blast loading
```

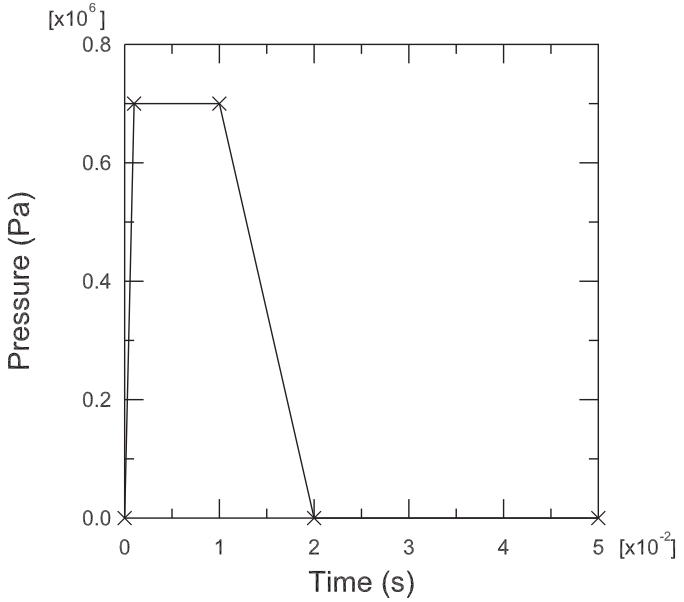


Figure 10–24 Pressure load as a function of time.

```
** Explicit analysis with a time duration of 50 ms
*DYNAMIC, EXPLICIT
, 50E-03
```

Applying the blast load

The *DLOAD option is used to apply the blast load to the plate. It is important to ensure that the pressure load is being applied in the correct direction. Positive pressure is defined as acting in the direction of the positive shell normal. For shell elements the positive normal direction is obtained using the right-hand rule about the nodes of the element, as shown in Figure 10–25. Since the magnitude of the load has been defined in the **BLAST** amplitude definition, we need to apply only a unit pressure under *DLOAD. This pressure is applied so that it pushes against the top of the plate (where the stiffeners are on the bottom of the plate). Such a pressure load will place the outer fibers of the stiffeners in tension. The full option is shown below:

```
*DLOAD, AMPLITUDE=BLAST
PLATE, P, 1.0
```

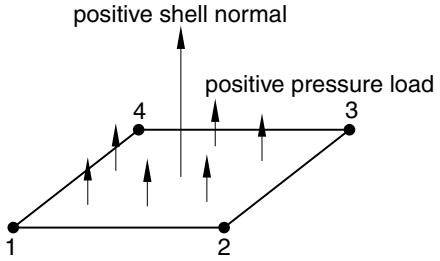


Figure 10–25 Definition of positive pressure load.

Output requests

To check on the progress of the solution, use the ∗MONITOR option to monitor the deflection at the center node of the plate during the analysis. In this example we monitor the out-of-plane displacement at the center node by adding the following command to the input file:

```
*MONITOR, NODE=<center node number>, DOF=2
```

For the input file shown in "Blast loading on a stiffened plate," Section A.9, the node number at the center of the plate is **411**.

Set the number of intervals during the step at which field data are written to the output database file (ODB) to 25. This ensures that the selected data outputs are written every 2 ms since the total time for the step is 50 ms. In general, you should try to limit the number of frames written during the analysis to keep the size of the output database file reasonable. In this analysis saving information every 2 ms should provide sufficient detail to study the response of the structure visually. This model requests field output for the stresses, plastic strains, and nodal displacements.

```
*OUTPUT, FIELD, NUMBER INTERVAL=25
*ELEMENT OUTPUT
S,PE
*NODE OUTPUT
U
```

A more detailed set of output can be saved for selected parts of the model by using the ∗OUTPUT, HISTORY option. Set the TIME INTERVAL parameter to 1.0E−4 seconds to write the required data at 500 points during the analysis. Write von Mises stress (MISES), equivalent plastic strain (PEEQ), and volumetric strain rate (ERV) for the elements in element set **STIFFMAX**. Since the nodes that will undergo the maximum displacements are at the center of the plate, use node set **NOUT** to output displacement and velocity history data for the center of the plate. In addition, save the following energy variables: kinetic energy (ALLKE), recoverable strain energy (ALLSE), work done (ALLWK), energy lost in plastic dissipation (ALLPD), total internal energy (ALLIE),

energy lost in viscous dissipation (ALLVD), artificial energy (ALLAE), and the energy balance (ETOTAL).

```
*OUTPUT, HISTORY, TIME INTERVAL=1.0E-4
*ELEMENT OUTPUT, ELSET=STIFFMAX
PEEQ, MISES
*NODE OUTPUT, NSET=NOUT
U, V
*ENERGY OUTPUT
ALLKE, ALLSE, ALLWK, ALLPD, ALLIE, ALLVD, ALLAE, ETOTAL
*END STEP
```

Save your input in a file called **blast_base.inp** since these results will serve as a base state from which to compare subsequent analyses. Run the analysis using the following command:

```
abaqus job=blast_base
```

10.5.6 Output

We now examine the output information contained in the status (**.sta**) file.

Status file

Information concerning model information, such as total mass and center of mass, and the initial stable time increment can be found at the top of the status file. The 10 most critical elements (i.e., those resulting in the smallest time increments) in rank order are also shown here. If your model contains a few elements that are much smaller than the rest of the elements in the model, the small elements will be the most critical elements and will control the stable time increment. The stable time increment information in the status file can indicate elements that are adversely affecting the stable time increment, allowing you to change the mesh to improve the situation, if necessary. It is ideal to have a mesh of roughly uniformly sized elements. In this example the mesh is uniform; thus, the 10 most critical elements share the same minimum time increment. The beginning of the status file is shown below.

```
-------------------------------------------------------------------------------
 MODEL INFORMATION (IN GLOBAL X-Y COORDINATES)
-------------------------------------------------------------------------------

   Total mass in model = 838.50
   Center of mass of model = ( 1.000000E+00, 3.488372E-03, 1.000000E+00)

    Moments of Inertia :
                About Center of Mass              About Origin
      I(XX)          2.849002E+02                 1.123410E+03
      I(YY)          5.519482E+02                 2.228948E+03
      I(ZZ)          2.712609E+02                 1.109771E+03
      I(XY)         -8.881784E-16                -2.925000E+00
      I(YZ)         -8.881784E-16                -2.925000E+00
      I(ZX)         -2.273737E-13                -8.385000E+02

-------------------------------------------------------------------------------
 STABLE TIME INCREMENT INFORMATION
```

```
------------------------------------------------------------------------------

  The stable time increment estimate for each element is based on
  linearization about the initial state.


   Initial time increment = 8.18646E-06

   Statistics for all elements:
      Mean = 1.30938E-05
      Standard deviation = 2.69043E-06

   Most critical elements :
    Element number    Rank    Time increment   Increment ratio
   ------------------------------------------------------------
         1022           1       8.186462E-06      1.000000E+00
         1024           2       8.186462E-06      1.000000E+00
         1027           3       8.186462E-06      1.000000E+00
         1029           4       8.186462E-06      1.000000E+00
         1033           5       8.186462E-06      1.000000E+00
         1038           6       8.186462E-06      1.000000E+00
         2022           7       8.186462E-06      1.000000E+00
         2024           8       8.186462E-06      1.000000E+00
         2027           9       8.186462E-06      1.000000E+00
         2029          10       8.186462E-06      1.000000E+00
```

During the analysis the status file can be viewed to monitor the progress of the analysis. Shown below is the beginning of the solution progress portion of the status file. Many more increments have been carried out than you would expect from an Abaqus/Standard analysis and that the output database file is being written at intervals of 2 ms.

```
------------------------------------------------------------------------------
 SOLUTION PROGRESS
------------------------------------------------------------------------------

 STEP 1  ORIGIN 0.0000

  Total memory used for step 1 is approximately 1.7 megabytes.
  Global time estimation algorithm will be used.
  Scaling factor:  1.0000
  Variable mass scaling factor at zero increment:  1.0000
               STEP      TOTAL         CPU       STABLE   CRITICAL      KINETIC
INCREMENT      TIME       TIME        TIME    INCREMENT    ELEMENT       ENERGY   MONITOR
       0  0.000E+00 0.000E+00   00:00:00 8.186E-06       1024   0.000E+00 0.000E+00
ODB Field Frame Number       0 of      25 requested intervals at increment zero.
ODB Field Frame Number       0 of       5 requested intervals at increment zero.
     244  2.005E-03 2.005E-03   00:00:00 8.182E-06       2035   4.499E+03 4.224E-03
ODB Field Frame Number       1 of      25 requested intervals at  2.005236E-03
     488  4.001E-03 4.001E-03   00:00:01 8.181E-06       2018   1.105E+04 2.506E-02
ODB Field Frame Number       2 of      25 requested intervals at  4.001393E-03
     733  6.005E-03 6.005E-03   00:00:01 8.138E-06       2030   5.879E+03 4.555E-02
ODB Field Frame Number       3 of      25 requested intervals at  6.004539E-03
     978  8.003E-03 8.003E-03   00:00:02 8.133E-06       2030   1.727E+02 4.976E-02
ODB Field Frame Number       4 of      25 requested intervals at  8.002752E-03
    1224  1.000E-02 1.000E-02   00:00:02 8.139E-06       2030   2.299E+03 4.461E-02
ODB Field Frame Number       5 of      25 requested intervals at  1.000000E-02
```

Output for the node referenced on the *MONITOR option is also included in this file.

10.5.7 Postprocessing

Run Abaqus/Viewer by entering the following command at the operating system prompt:

```
abaqus viewer odb=blast_base
```

Changing the view

The default view is isometric, which does not provide a particularly clear view of the plate. To improve the viewpoint, rotate the view using the options in the **View** menu or the tools in the **View Manipulation** toolbar. Specify the view and select the viewpoint method for rotating the view. Enter the *X*-, *Y*-, and *Z*-coordinates of the viewpoint vector as `1,0.5,1` and the coordinates of the up vector as `0,1,0`.

Verifying shell section assignment

You can also visualize the section assignment and the shell thickness while postprocessing the results. For example, regions with common section assignments can be color coded to verify that the properties were assigned correctly (select **Sections** from the **Color Code** toolbar to color the mesh according to section assignment). To render the shell thickness, select **View→ODB Display Options** from the main menu bar. In the **ODB Display Options** dialog box, toggle on **Render shell thickness** and click **Apply**. If the model looks correct, as shown in Figure 10–26, toggle off this option and click **OK** before proceeding with the rest of the postprocessing instructions. Otherwise, correct the section assignment and rerun the job.

Animation of results

As noted in earlier examples, animating your results will provide a general understanding of the dynamic response of the plate under the blast loading. First, plot the deformed model shape. Then, create a time-history animation of the deformed shape. Use the **Animation Options** dialog box to change the mode to **Play once**.

You will see from the animation that as the blast loading is applied, the plate begins to deflect. Over the duration of the load the plate begins to vibrate and continues to vibrate after the blast load has dropped to zero. The maximum displacement occurs at approximately 8 ms, and a displaced plot of that state is shown in Figure 10–27. The animation images can be saved to a file for playback at a later time.

To save the animation:

1. From the main menu bar, select **Animate→Save As**.

   The **Save Image Animation** dialog box appears.

2. In the **Settings** field, enter the file name `blast_base`.

   The format of the animation can be specified as AVI, QuickTime, VRML, or Compressed VRML.

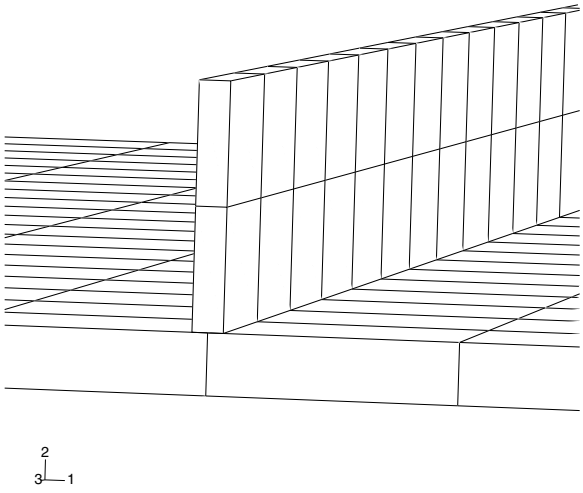


Figure 10–26 Plate with shell thickness displayed.

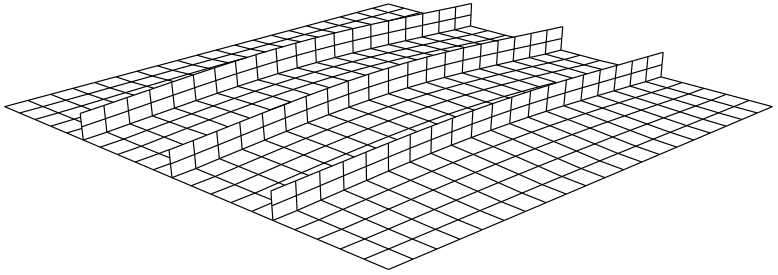


Figure 10–27 Displaced shape at 8 ms.

3. Choose the **QuickTime** format, and click **OK**.

   The animation is saved as **`blast_base.mov`** in your current directory. Once saved, your animation can be played external to Abaqus/Viewer using industry-standard animation software.

History output

Since it is not easy to see the deformation of the plate from the deformed plot, it is desirable to view the deflection response of the central node in the form of a graph. The displacement of the node in the center of the plate is of particular interest since the largest deflection occurs at this node.

Display the displacement history of the central node, as shown in Figure 10–28 (with displacements in millimeters).

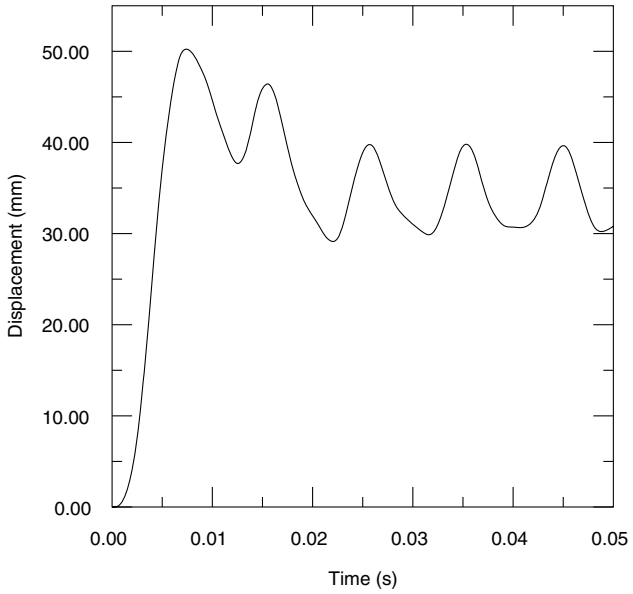


Figure 10–28 Central node displacement as a function of time.

To generate a history plot of the central node displacement:

1. In the Results Tree, double-click the history output data named **`Spatial displacement: U2`** at the node in the center of the plate (set **`NOUT`**).
2. Save the current *X*–*Y* data: in the Results Tree, click mouse button 3 on the data name and select **Save As** from the menu that appears. Name the data **`DISP`**.

   The units of the displacements in this plot are meters. Modify the data to create a plot of displacement (in millimeters) versus time by creating a new data object.
3. In the Results Tree, expand the **XYData** container.

   The **`DISP`** data are listed underneath.
4. In the Results Tree, double-click **XYData**; then select **Operate on XY data** in the **Create XY Data** dialog box. Click **Continue**.
5. In the **Operate on XY Data** dialog box, multiply **`DISP`** by 1000 to create the plot with the displacement values in millimeters instead of meters. The expression at the top of the dialog box should appear as:

```
"DISP" * 1000
```

6. Click **Plot Expression** to see the modified *X*–*Y* data. Save the data as **`U2_BASE`**.
7. Close the **Operate on XY Data** dialog box.
8. Click the **Axis Options** tool in the toolbox. In the **Axis Options** dialog box, change the *X*-axis title to **`Time (s)`** and the *Y*-axis title to **`Displacement (mm)`**. Click **OK** to close the dialog box. The resulting plot is shown in Figure 10–28.

   The plot shows that the displacement reaches a maximum of 50.2 mm at 7.7 ms and then oscillates after the blast load is removed.

The other quantities saved as history output in the output database are the total energies of the model. The energy histories can help identify possible shortcomings in the model as well as highlight significant physical effects. Display the histories of five different energy output variables—ALLAE, ALLIE, ALLKE, ALLPD, and ALLSE.

To generate history plots of the model energies:

1. Save the history results for the ALLAE, ALLIE, ALLKE, ALLPD, and ALLSE output variables as *X*–*Y* data. A default name is given to each curve; rename each according to its output variable name: **`ALLAE`**, **`ALLKE`**, etc.
2. In the Results Tree, expand the **XYData** container.

   The **`ALLAE`**, **`ALLIE`**, **`ALLKE`**, **`ALLPD`**, and **`ALLSE`** *X*–*Y* data objects are listed underneath.
3. Select **`ALLAE`**, **`ALLIE`**, **`ALLKE`**, **`ALLPD`**, and **`ALLSE`** using [Ctrl]+Click; click mouse button 3, and select **Plot** from the menu that appears to plot the energy curves.
4. To more clearly distinguish between the different curves in the plot, open the **Curve Options** dialog box and change their line styles.
   - For the curve **`ALLSE`**, select a dashed line style.
   - For the curve **`ALLPD`**, select a dotted line style.
   - For the curve **`ALLAE`**, select a chain dashed line style.
   - For the curve **`ALLIE`**, select the second thinnest line type.
5. To change the position of the legend, open the **Chart Legend Options** dialog box and switch to the **Area** tabbed page.
6. In the **Position** region of this page, toggle on **Inset** and click **Dismiss**. Drag the legend in the viewport so that it fits within the grid, as shown in Figure 10–29.

We can see that once the load has been removed and the plate vibrates freely, the kinetic energy increases as the strain energy decreases. When the plate is at its maximum deflection and, therefore, has its maximum strain energy, it is almost entirely at rest, causing the kinetic energy to be at a minimum.

The plastic strain energy rises to a plateau and then rises again. From the plot of kinetic energy we can see that the second rise in plastic strain energy occurs when the plate has rebounded from

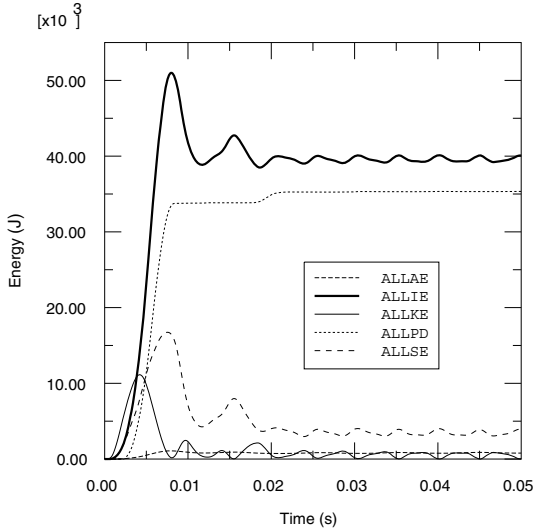


Figure 10–29 Energy quantities as a function of time.

its maximum displacement and is moving back in the opposite direction. We are, therefore, seeing plastic deformation on the rebound after the blast pulse.

Even though there is no indication that hourglassing is a problem in this analysis, study the artificial strain energy to make sure. As discussed in Chapter 4, "Using Continuum Elements," artificial strain energy or "hourglass stiffness" is the energy used to control hourglass deformation, and the output variable ALLAE is the accumulated artificial strain energy. This discussion on hourglass control applies equally to shell elements. Since energy is dissipated as plastic deformation as the plate deforms, the total internal energy is much greater than the elastic strain energy alone. Therefore, it is most meaningful in this analysis to compare the artificial strain energy to an energy quantity that includes the dissipated energy as well as the elastic strain energy. Such a variable is the total internal energy, ALLIE, which is a summation of all internal energy quantities. The artificial strain energy is approximately 2% of the total internal energy, indicating that hourglassing is not a problem.

One thing we can notice from the deformed shape is that the central stiffener is subject to almost pure in-plane bending. Using only two first-order, reduced-integration elements through the depth of the stiffener is not sufficient to model in-plane bending behavior. While the solution from this coarse mesh appears to be adequate since there is little hourglassing, for completeness we will study how the solution changes when we refine the mesh of the stiffener. Use caution when you refine the mesh, since mesh refinement will increase the solution time by increasing the number of elements and decreasing the element size.

An input file for a model with a refined stiffener mesh is included in "Blast loading on a stiffened plate," Section A.9 (**`blast_refined.inp`**); four elements through the depth are used

to model the stiffener instead of two. This increase in the number of elements increases the solution time by approximately 20%. In addition, the stable time increment decreases by approximately a factor of two as a result of the reduction of the smallest element dimension in the stiffeners. Since the total increase in solution time is a combination of the two effects, the solution time of the refined mesh increases by approximately a factor of 1.2×2, or 2.4, over that of the original mesh.

Figure 10–30 shows the histories of artificial energy for both the original mesh and the mesh with the refined stiffeners. The artificial energy is slightly lower in the refined mesh. As a result, we would not expect the results to change significantly from the original to the refined mesh.

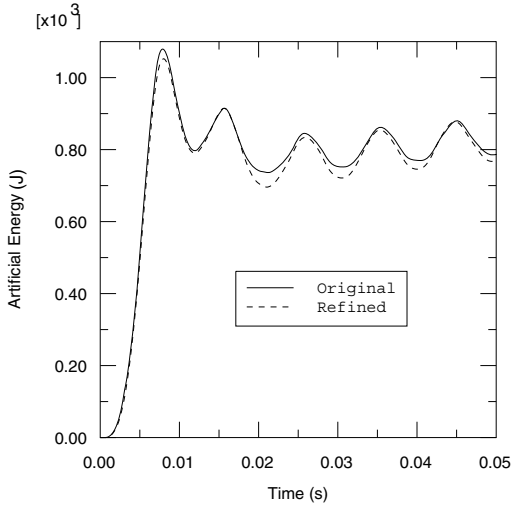


Figure 10–30 Artificial energy in the original and refined meshes.

Figure 10–31 shows that the displacement of the plate's central node is almost identical in both cases, indicating that the original mesh is capturing the overall response adequately. One advantage of the refined mesh, however, is that it better captures the variation of stress and plastic strain through the stiffeners.

Contour plots

In this section you will use the contour plotting capability of Abaqus/Viewer to display the von Mises stress and equivalent plastic strain distributions in the plate. Use the model with the refined stiffener mesh to create the plots.

To generate contour plots of the von Mises stress and equivalent plastic strain:

1. From the list of variable types on the left side of the **Field Output** toolbar, select **Primary**.

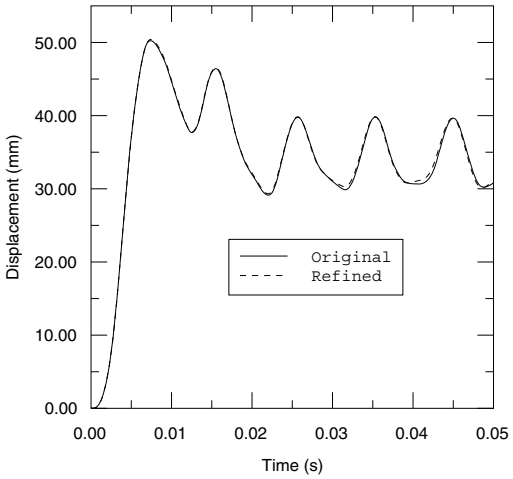


Figure 10–31 Central node displacement history for the original and refined meshes.

2. From the list of output variables in the center of the toolbar, select **S**. The stress invariants and components are available in the next list to the right. Select the **Mises** stress invariant.
3. From the main menu bar, select **Result→Section Points**.
4. In the **Section Points** dialog box that appears, select **Top and bottom** as the active locations and click **OK**.
5. Select **Plot→Contours→On Deformed Shape**, or use the tool from the toolbox.

   Abaqus plots the contours of the von Mises stress on both the top and bottom faces of each shell element. To see this more clearly, rotate the model in the viewport.

   The view that you set earlier for the animation exercise should be changed so that the stress distribution is clearer.
6. Change the view back to the default isometric view using the

tool in the **Views** toolbar.

   Tip: If the **Views** toolbar is not visible, select **View→Toolbars→Views** from the main menu bar.

   Figure 10–32 shows a contour plot of the von Mises stress at the end of the analysis.
7. Similarly, contour the equivalent plastic strain. Select **Primary** from the list of variable types on the left side of the **Field Output** toolbar and select **PEEQ** from the list of output variables next to it.

   Figure 10–33 shows a contour plot of the equivalent plastic strain at the end of the analysis.

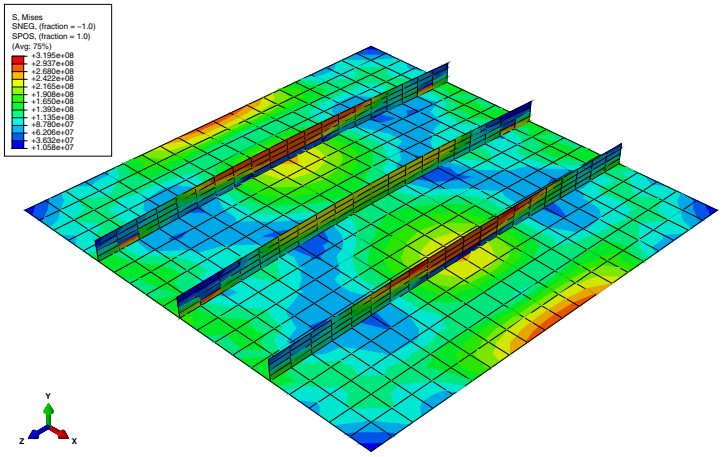


Figure 10–32 Contour plot of von Mises stress at 50 ms.

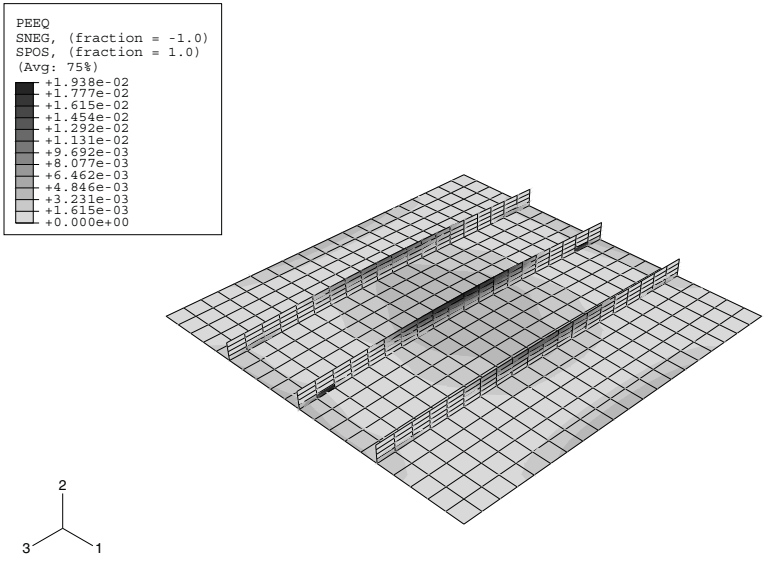


Figure 10–33 Contour plot of equivalent plastic strain at 50 ms.

10.5.8 Reviewing the analysis

The objective of this analysis is to study the deformation of the plate and the stress in various parts of the structure when it is subjected to a blast load. To judge the accuracy of the analysis, you will need to consider the assumptions and approximations made and identify some of the limitations of the model.

Damping

Undamped structures continue to vibrate with constant amplitude. Over the 50 ms of this simulation, the frequency of the oscillation can be seen to be approximately 100 Hz. A constant amplitude vibration is not the response that would be expected in practice since the vibrations in this type of structure would tend to die out over time and effectively disappear after 5–10 oscillations. The energy loss typically occurs by a variety of mechanisms including frictional effects at the supports and damping by the air.

Consequently, we need to consider the presence of damping in the analysis to model this energy loss. The energy dissipated by viscous effects, ALLVD, is nonzero in the analysis, indicating that there is already some damping present. By default, a *bulk viscosity* damping (discussed in Chapter 9, "Nonlinear Explicit Dynamics") is always present and is introduced to improve the modeling of high-speed events.

In this shell model only linear damping is present. With the default value the oscillations would eventually die away, but it would take a long time because the bulk viscosity damping is very small. Material damping should be used to introduce a more realistic structural response. Modify the material data block to include damping, setting the mass proportional damping to 50.0.

```
*DAMPING, ALPHA=50.0, BETA=0.0
```

BETA is the parameter that controls stiffness proportional damping, and at this stage we will leave it set to zero.

The duration of the oscillation of the plate is approximately 30 ms, so we need to increase the analysis period to allow enough time for the vibration to be damped out. Therefore, increase the analysis period to 150 ms.

The results of the damped analysis clearly show the effect of mass proportional damping. Figure 10–34 shows the displacement history of the central node for both the damped and undamped analyses. (We have extended the analysis time to 150 ms for the undamped model to compare the data more effectively.) The peak response is also reduced due to damping. By the end of the damped analysis the oscillation has decayed to a nearly static condition.

Rate dependence

Some materials, such as mild steel, show an increase in the yield stress with increasing strain rate. In this example the loading rate is high, so strain-rate dependence is likely to be important. The ∗RATE DEPENDENT option is used with the ∗PLASTIC option to introduce strain-rate dependence.

Add the following to the ∗MATERIAL option block under the ∗PLASTIC option:

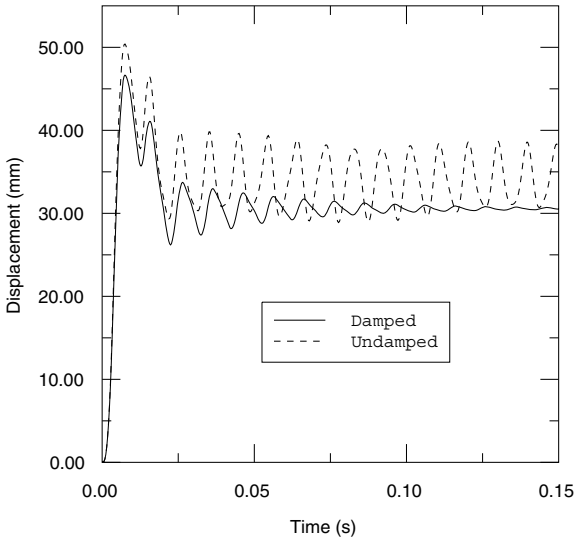


Figure 10–34 Damped and undamped displacement histories.

```
*RATE DEPENDENT
40.0, 5.0
```

With this definition of rate-dependent behavior, the ratio of the dynamic yield stress to the static yield stress (R) is given for an equivalent plastic strain rate ($\dot{\bar{\varepsilon}}^{pl}$), according to the equation $\dot{\bar{\varepsilon}}^{pl} = D\,(R-1)^n$, where D and n are material constants (40 and 5 in this case).

When the *RATE DEPENDENT option is included, the yield stress effectively increases as the strain rate increases. Therefore, because the elastic modulus is higher than the plastic modulus, we expect a stiffer response in the analysis with rate dependence. Both the displacement history of the central portion of the plate shown in Figure 10–35 and the history of plastic strain shown in Figure 10–36 confirm that the response is indeed stiffer when rate dependence is included.

10.6 Hyperelasticity

We now turn our attention to another class of material nonlinearity, namely, the nonlinear elastic response exhibited by rubber materials.

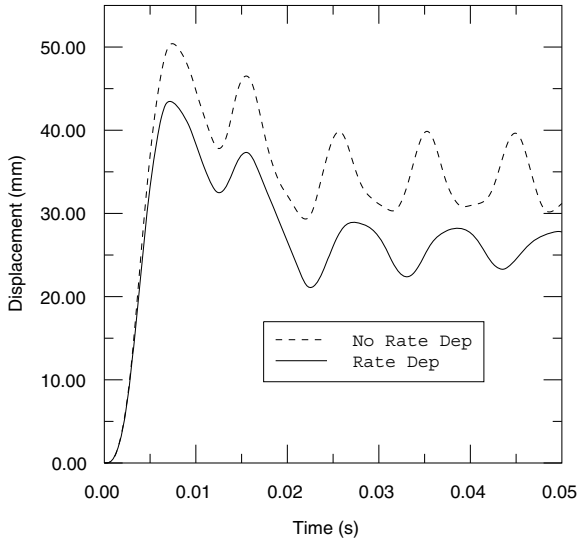


Figure 10–35 Displacement of the central node with and without rate dependence.

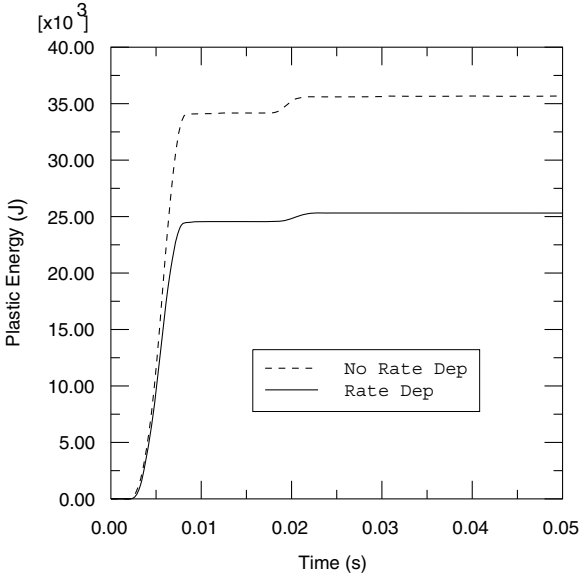


Figure 10–36 Plastic strain energy with and without rate dependence.

10.6.1 Introduction

The stress-strain behavior of typical rubber materials, shown in Figure 10–37, is elastic but highly nonlinear.

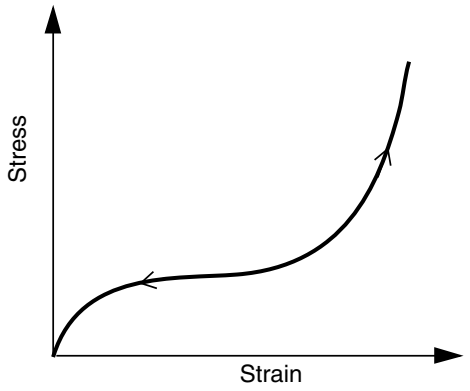


Figure 10–37 Typical stress-strain curve for rubber.

This type of material behavior is called *hyperelasticity*. The deformation of hyperelastic materials, such as rubber, remains elastic up to large strain values (often well over 100%).

Abaqus makes the following assumptions when modeling a hyperelastic material:

- The material behavior is elastic.
- The material behavior is isotropic.
- The simulation will include nonlinear geometric effects (NLGEOM=YES will be used).

In addition, Abaqus/Standard assumes the hyperelastic material is incompressible by default. Abaqus/Explicit assumes the material is nearly incompressible (Poisson's ratio is 0.475 by default).

Elastomeric foams are another class of highly nonlinear, elastic materials. They differ from rubber materials in that they have very compressible behavior when subjected to compressive loads. They are modeled with a separate material model in Abaqus and are not discussed in detail in this guide.

10.6.2 Compressibility

Most solid rubber materials have very little compressibility compared to their shear flexibility. This behavior is not a problem with plane stress, shell, or membrane elements. However, it can be a problem when using other elements, such as plane strain, axisymmetric, and three-dimensional solid elements. For example, in applications where the material is not highly confined, it would be quite satisfactory to assume that the material is fully incompressible: the volume of the material cannot change except for

thermal expansion. In cases where the material is highly confined (such as an O-ring used as a seal), the compressibility must be modeled correctly to obtain accurate results.

Abaqus/Standard has a special family of "hybrid" elements that must be used to model the fully incompressible behavior seen in hyperelastic materials. These "hybrid" elements are identified by the letter 'H' in their name; for example, the hybrid form of the 8-node brick, C3D8, is called C3D8H.

Except for plane stress and uniaxial cases, it is not possible to assume that the material is fully incompressible in Abaqus/Explicit because the program has no mechanism for imposing such a constraint at each material calculation point. An incompressible material also has an infinite wave speed, resulting in a time increment of zero. Therefore, we must provide some compressibility. The difficulty is that, in many cases, the actual material behavior provides too little compressibility for the algorithms to work efficiently. Thus, except for plane stress and uniaxial cases, the user must provide enough compressibility for the code to work, knowing that this makes the bulk behavior of the model softer than that of the actual material. Some judgment is, therefore, required to decide whether or not the solution is sufficiently accurate, or whether the problem can be modeled at all with Abaqus/Explicit because of this numerical limitation. We can assess the relative compressibility of a material by the ratio of its initial bulk modulus, K_0, to its initial shear modulus, μ_0. Poisson's ratio, ν, also provides a measure of compressibility since it is defined as

$$\nu = \frac{3\left(K_0/\mu_0\right) - 2}{6\left(K_0/\mu_0\right) + 2}.$$

Table 10–2 provides some representative values.

Table 10–2 Relationship between compressibility and Poisson's ratio.

K_0/μ_0	**Poisson's ratio**
10	0.452
20	0.475
50	0.490
100	0.495
1,000	0.4995
10,000	0.49995

If no value is given for the material compressibility, by default Abaqus/Explicit assumes $K_0/\mu_0 = 20$, corresponding to Poisson's ratio of 0.475. Since typical unfilled elastomers have K_0/μ_0 ratios in the range of 1,000 to 10,000 ($\nu = 0.4995$ to $\nu = 0.49995$) and filled elastomers have K_0/μ_0 ratios in the range of 50 to 200 ($\nu = 0.490$ to $\nu = 0.497$), this default provides much more compressibility than is available in most elastomers. However, if the elastomer is relatively unconfined, this softer modeling of the material's bulk behavior usually provides quite accurate results. Unfortunately, in cases where the material is highly confined—such as when it is in contact with stiff, metal parts and has a very small

amount of free surface, especially when the loading is highly compressive—it may not be feasible to obtain accurate results with Abaqus/Explicit.

If you are defining the compressibility rather than accepting the default value in Abaqus/Explicit, an upper limit of 100 is suggested for the ratio of K_0/μ_0. Larger ratios introduce high frequency noise into the dynamic solution and require the use of excessively small time increments.

10.6.3 Strain energy potential

Abaqus uses a *strain energy potential* (*U*), rather than a Young's modulus and Poisson's ratio, to relate stresses to strains in hyperelastic materials. Several different strain energy potentials are available: a polynomial model, the Ogden model, the Arruda-Boyce model, the Marlow model, and the van der Waals model. Simpler forms of the polynomial model are also available, including the Mooney-Rivlin, neo-Hookean, reduced polynomial, and Yeoh models.

The polynomial form of the strain energy potential is one that is commonly used. Its form is

$$U = \sum_{i+j=1}^{N} C_{ij}(\bar{I}_1 - 3)^i(\bar{I}_2 - 3)^j + \sum_{i=1}^{N} \frac{1}{D_i}(J_{el} - 1)^{2i},$$

where U is the strain energy potential; J_{el} is the elastic volume ratio; $\bar{I}_1$ and $\bar{I}_2$ are measures of the distortion in the material; and N, C_{ij}, and D_i are material parameters, which may be functions of temperature. The C_{ij} parameters describe the shear behavior of the material, and the D_i parameters introduce compressibility. If the material is fully incompressible (a condition not allowed in Abaqus/Explicit), all the values of D_i are set to zero and the second part of the equation shown above can be ignored. If the number of terms, N, is one, the initial shear modulus, μ_0, and bulk modulus, K_0, are given by

$$\mu_0 = 2(C_{01} + C_{10}),$$

and

$$K_0 = \frac{2}{D_1}.$$

If the material is also incompressible, the equation for the strain energy density is

$$U = C_{10}(\bar{I}_1 - 3) + C_{01}(\bar{I}_2 - 3).$$

This expression is commonly referred to as the *Mooney-Rivlin* material model. If C_{01} is also zero, the material is called *neo-Hookean*.

The other hyperelastic models are similar in concept and are described in "Hyperelasticity," Section 22.5 of the Abaqus Analysis User's Manual.

You must provide Abaqus with the relevant material parameters to use a hyperelastic material. For the polynomial form these are C_{ij} and D_i. It is possible that you will be supplied with these parameters when modeling hyperelastic materials; however, more likely you will be given test data for

the materials that you must model. Fortunately, Abaqus can accept test data directly and calculate the material parameters for you (using a least squares fit).

10.6.4 Defining hyperelastic behavior using test data

A convenient way of defining a hyperelastic material is to supply Abaqus with experimental test data. Abaqus then calculates the constants using a least squares method. Abaqus can fit data for the following experimental tests:

- Uniaxial tension and compression
- Equibiaxial tension and compression
- Planar tension and compression (pure shear)
- Volumetric tension and compression

The deformation modes seen in these tests and the Abaqus input options used to define the data for each are shown in Figure 10–38. Unlike plasticity data, the test data for hyperelastic materials must be given to Abaqus as nominal stress and nominal strain values.

Volumetric compression data only need to be given if the material's compressibility is important. Normally in Abaqus/Standard it is not important, and the default fully incompressible behavior is used. As noted earlier, Abaqus/Explicit assumes a small amount of compressibility if no volumetric test data are given.

Obtaining the best material model from your data

The quality of the results from a simulation using hyperelastic materials strongly depends on the material test data that you provide Abaqus. Typical tests are shown in Figure 10–38. There are several things that you can do to help Abaqus calculate the best possible material parameters.

Wherever possible, try to obtain experimental test data from more than one deformation state—this allows Abaqus to form a more accurate and stable material model. However, some of the tests shown in Figure 10–38 produce equivalent deformation modes for incompressible materials. The following are equivalent tests for incompressible materials:

- Uniaxial tension ↔ Equibiaxial compression
- Uniaxial compression ↔ Equibiaxial tension
- Planar tension ↔ Planar compression

You do not need to include data from a particular test if you already have data from another test that models a particular deformation mode.

In addition, the following may improve your hyperelastic material model:

- Obtain test data for the deformation modes that are likely to occur in your simulation. For example, if your component is loaded in compression, make sure that your test data include compressive, rather than tensile, loading.

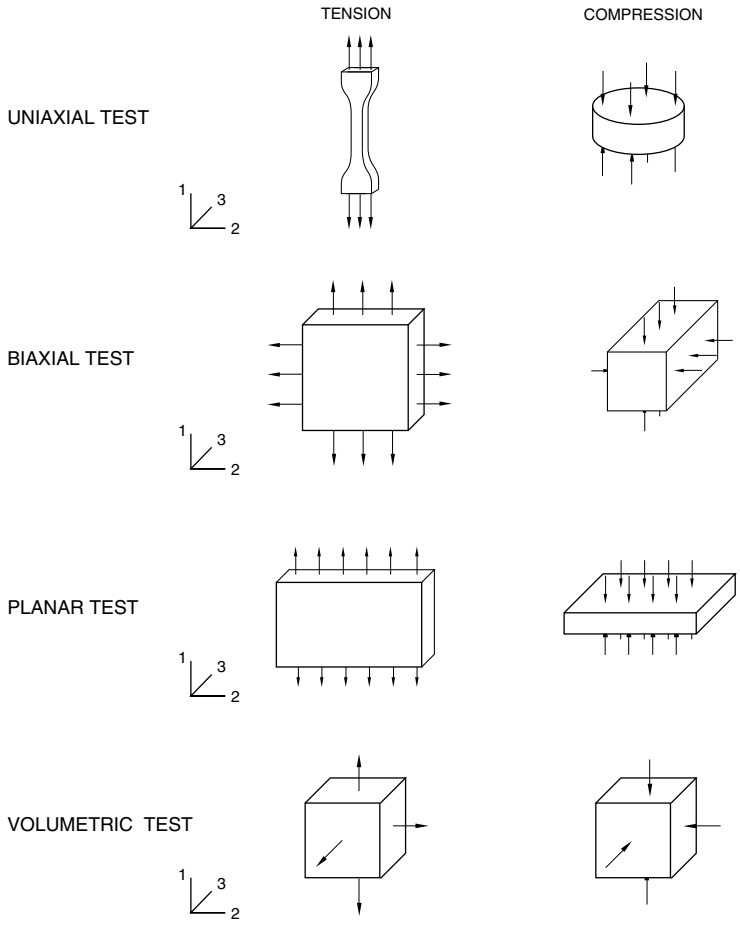


Figure 10–38 Deformation modes and Abaqus input options for the various experimental tests for defining hyperelastic material behavior.

- Both tension and compression data are allowed, with compressive stresses and strains entered as negative values. If possible, use compression or tension data depending on the application, since the fit of a single material model to both tensile and compressive data will normally be less accurate than for each individual test.
- Try to include test data from the planar test. This test measures shear behavior, which can be very important.

- Provide more data at the strain magnitudes that you expect the material will be subjected to during the simulation. For example, if the material will only have small tensile strains, say under 50%, do not provide much, if any, test data at high strain values (over 100%).
- Perform one-element simulations of the experimental tests and compare the results Abaqus calculates to the experimental data. If the computational results are poor for a particular deformation mode that is important to you, try to obtain more experimental data for that deformation mode. These one-element simulations are very easy to perform in Abaqus/CAE. Please consult the Abaqus/CAE User's Manual for details.

Stability of the material model

It is common for the material model determined from the test data to be unstable at certain strain magnitudes. Abaqus performs a stability check to determine the strain magnitudes where unstable behavior will occur and prints a warning message in the data (`.dat`) file. You should check this information carefully since your simulation may not converge if any part of the model experiences strains beyond the stability limits. The stability checks are done for specific deformations, so it is possible for the material to be unstable at the strain levels indicated if the deformation is more complex. Likewise, it is possible for the material to become unstable at lower strain levels if the deformation is more complex. In Abaqus/Standard your simulation may not converge if a part of the model exceeds the stability limits.

See "Hyperelastic behavior of rubberlike materials," Section 22.5.1 of the Abaqus Analysis User's Manual, for suggestions on improving the accuracy and stability of the test data fit.

10.7 Example: axisymmetric mount

You have been asked to find the axial stiffness of the rubber mount shown in Figure 10–39 and to identify any areas of high maximum principal stress that might limit the fatigue life of the mount. The mount is bonded at both ends to steel plates. It will experience axial loads up to 5.5 kN distributed uniformly across the plates. The cross-section geometry and dimensions are given in Figure 10–39.

You can use axisymmetric elements for this simulation since both the geometry of the structure and the loading are axisymmetric. Therefore, you only need to model a plane through the component: each element represents a complete 360° ring. You will examine the static response of the mount; therefore, you will use Abaqus/Standard for your analysis.

10.7.1 Symmetry

You do not need to model the whole section of this axisymmetric component because the problem is symmetric about a horizontal line through the center of the mount. By modeling only half of the section, you can use half as many elements and, hence, approximately half the number of degrees of freedom.

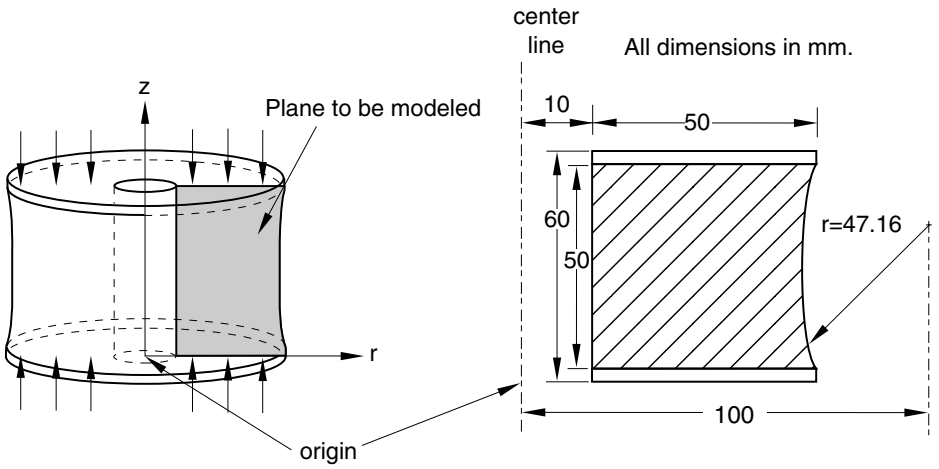


Figure 10–39 Axisymmetric mount.

This significantly reduces the run time and storage requirements for the analysis or, alternatively, allows you to use a more refined mesh.

Many problems contain some degree of symmetry. For example, mirror symmetry, cyclic symmetry, axisymmetry, or repetitive symmetry (shown in Figure 10–40) are common. More than one type of symmetry may be present in the structure or component that you want to model.

When modeling just a portion of a symmetric component, you have to add boundary conditions to make the model behave as if the whole component were being modeled. You may also have to adjust the applied loads to reflect the portion of the structure actually being modeled. Consider the portal frame in Figure 10–41.

The frame is symmetric about the vertical line shown in the figure. To maintain symmetry in the model, any nodes on the symmetry line must be constrained from translating in the 1-direction and from rotating about the 2- or 3-axes (degrees of freedom 5 and 6). Therefore, the symmetry constraints are

```
*BOUNDARY
<node>,1
<node>,5,6
```

In the frame problem the load is applied along the model's symmetry plane; therefore, only half of the total value should be applied to the portion you are modeling.

In axisymmetric analyses using axisymmetric elements, such as this rubber mount example, we need model only the cross-section of the component. The element formulation automatically includes the effects of axial symmetry.

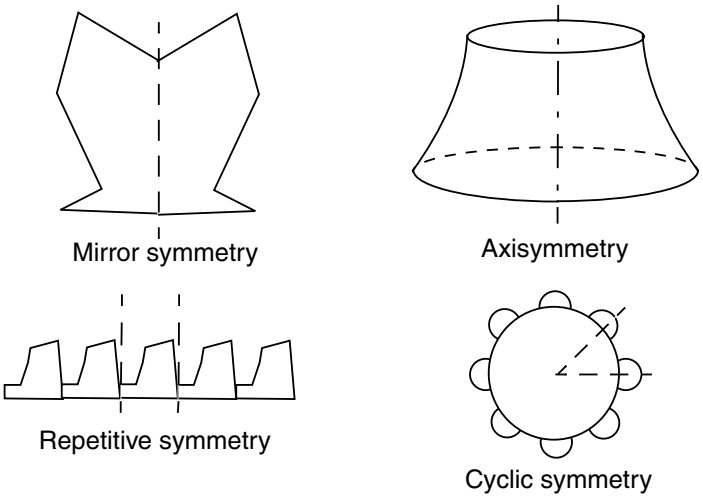


Figure 10–40 Various forms of symmetry.

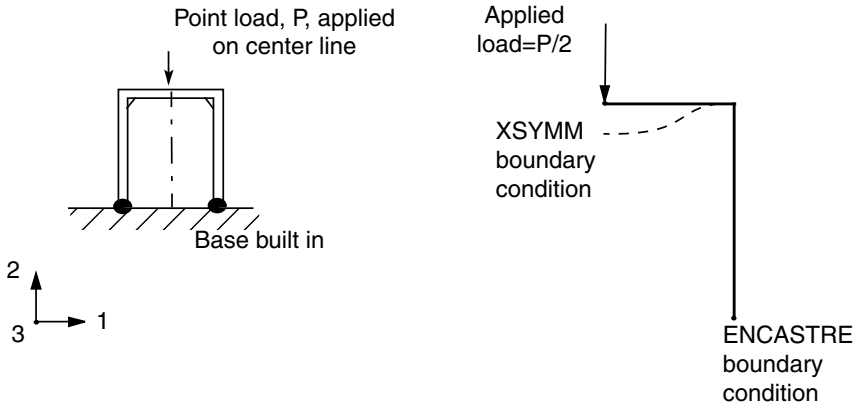


Figure 10–41 Symmetric portal frame.

10.7.2 Coordinate system

The model in this example uses the default *r–z* (1–2) axisymmetric coordinate system in this simulation. Its origin is placed at the level of the bottom of the plate, as shown in Figure 10–39.

10.7.3 Mesh design

The mesh in this example uses a 30 × 15 mesh of first-order, axisymmetric, hybrid solid elements (CAX4H) for the rubber mount. Only the bottom half of the mount is specified in the model, as shown in Figure 10–42.

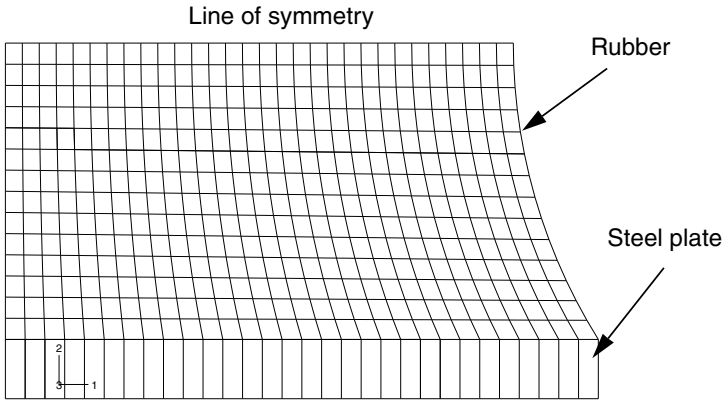


Figure 10–42 Mesh for the rubber mount.

Hybrid elements are required in this example because the material is fully incompressible. The elements are not expected to be subjected to bending, so shear locking in these fully integrated elements should not be a concern. Model the steel plates with a single layer of incompatible mode elements (CAX4I) because it is possible that the plates may bend as the rubber underneath them deforms.

The node and element numbers from the input file given in "Axisymmetric mount," Section A.10, are shown in Figure 10–43 and Figure 10–44. These will be used in the discussion of this example. If you build the model yourself, it will probably have different node and element numbers.

10.7.4 Preprocessing—creating the model

The steps that follow assume that you have access to the full input file for this example. This input file, **`mount.inp`**, is provided in "Blast loading on a stiffened plate," Section A.9, in the online HTML version of this manual. Instructions on how to fetch and run the script are given in Appendix A, "Example Files."

If you use Abaqus/CAE or another preprocessor to create the mesh for this model, try to create a node set **`MIDDLE`** containing all the nodes on the symmetry plane, and apply a pressure load of 0.50 MPa to the bottom of the plate (Figure 10–45). Check that this pressure results in a total applied load of 5.5 kN (5.5 kN = 0.50 MPa × $\pi \left(r_o^2 - r_i^2\right)$). If you can't create the mesh, the Abaqus input options used to create the model can be found in "Axisymmetric mount," Section A.10.

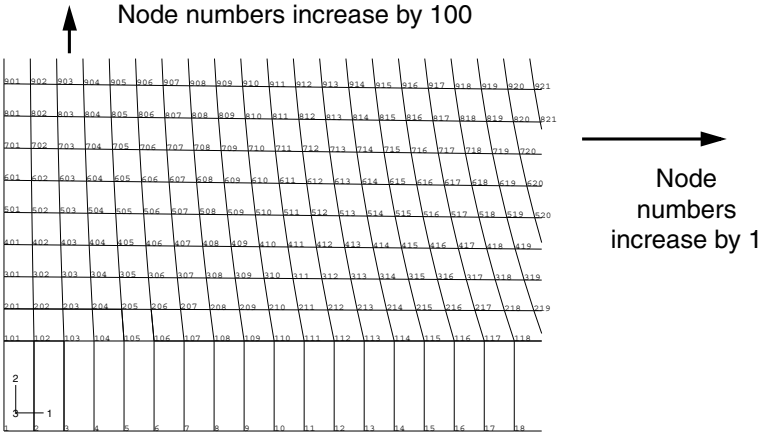


Figure 10–43 Node numbers.

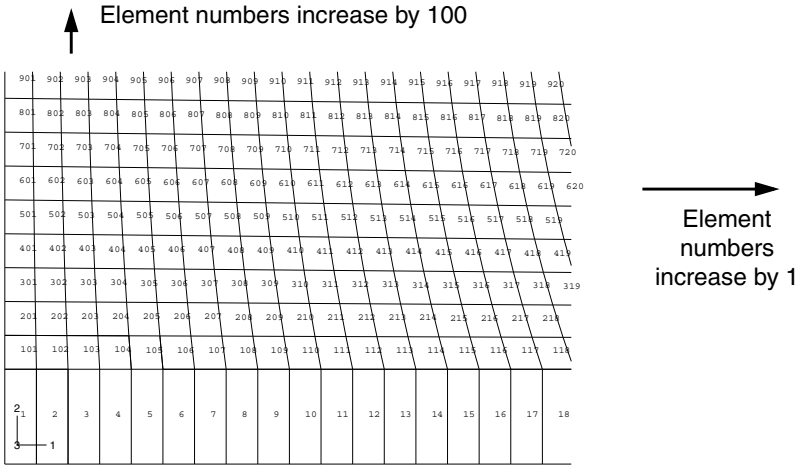


Figure 10–44 Element numbers.

If you wish to create the entire model using Abaqus/CAE, refer to "Example: axisymmetric mount," Section 10.7 of Getting Started with Abaqus: Interactive Edition.

10.7.5 Reviewing the input file—the model data

We review the model data, including the geometry definition (nodes and elements) and the material properties.

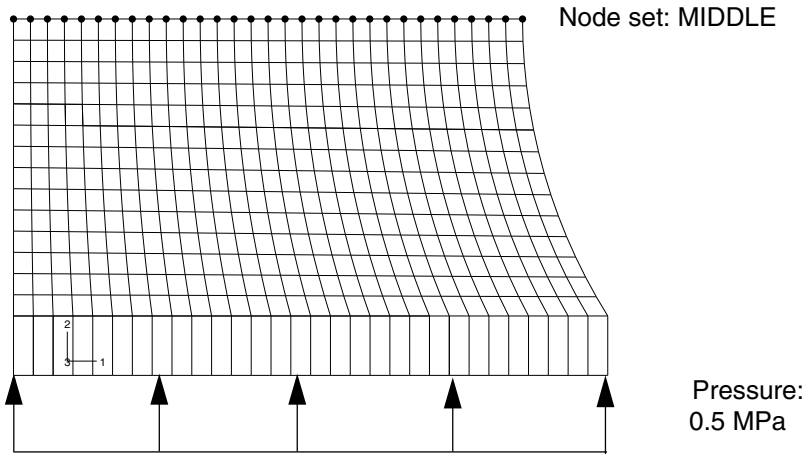


Figure 10–45 Node set **MIDDLE** and pressure loading.

Model description

The input file should contain a suitable description of the analysis.

```
*HEADING
Axisymmetric mount analysis under axial loading
S.I. Units (m, kg, N, sec)
```

Nodal coordinates and element connectivity

There will be at least two *ELEMENT option blocks in the input file since two different types of elements are used in the simulation. It is a good idea to check that the element types are correct and that the element sets containing the elements have descriptive names. The *ELEMENT options in your input file should look like

```
*ELEMENT, TYPE=CAX4I, ELSET=PLATE
*ELEMENT, TYPE=CAX4H, ELSET=RUBBER
```

Node sets

Check that the node set **MIDDLE** has been created. If it has not, add it using an editor.

Property definition

Two element property definitions are required: one for the elements modeling the rubber and one for those modeling the plates. The following element property definitions should be in your model:

```
*SOLID SECTION, MATERIAL=RUBBER, ELSET=RUBBER
*SOLID SECTION, MATERIAL=STEEL, ELSET=PLATE
```

Material properties: hyperelastic model for the rubber

You have been given some experimental test data, shown in Figure 10–46, for the rubber material used in the mount. Three different sets of test data—a uniaxial test, a biaxial test, and a planar (shear) test—are available. You decide to have Abaqus calculate the appropriate hyperelastic material constants from the test data. You are not sure how large the strains will be in the rubber mount, but you suspect that they will be under 2.0.

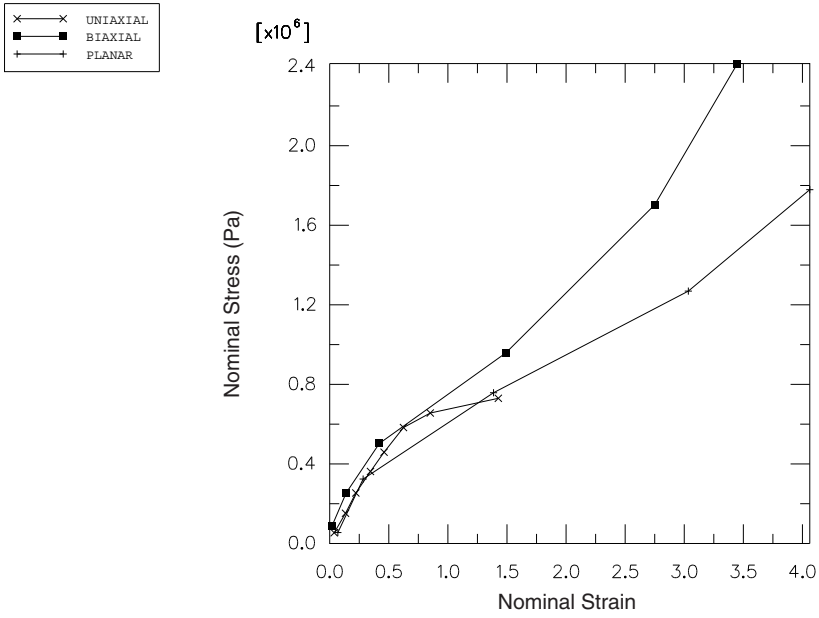


Figure 10–46 Material test data for the rubber material.

The test data for the biaxial and planar tests go well beyond this magnitude, so you decide to perform a one-element simulation of the experimental tests to confirm that the coefficients that Abaqus calculates from the test data are adequate.

Use a first-order, polynomial strain energy function to model the rubber material. Indicate these choices by using the N=1 and POLYNOMIAL parameters on the *HYPERELASTIC option. Use the TEST DATA INPUT parameter to indicate that Abaqus should find the material constants from the test data you will provide. The test data are given on options that immediately follow the *HYPERELASTIC option. The data should be entered as nominal stress and the corresponding nominal strain, with negative values indicating compression. You may be able to enter the data directly using your preprocessor (for instance, if you are using Abaqus/CAE); otherwise, you will have to add it to your input file with an editor. The material definition for the rubber will look like

```
*MATERIAL, NAME=RUBBER
*HYPERELASTIC, N=1, POLYNOMIAL, TEST DATA INPUT
*UNIAXIAL TEST DATA
 0.054E6,  0.0380
 0.152E6,  0.1338
 0.254E6,  0.2210
 0.362E6,  0.3450
 0.459E6,  0.4600
 0.583E6,  0.6242
 0.656E6,  0.8510
 0.730E6,  1.4268
*BIAXIAL TEST DATA
 0.089E6,  0.0200
 0.255E6,  0.1400
 0.503E6,  0.4200
 0.958E6,  1.4900
 1.703E6,  2.7500
 2.413E6,  3.4500
*PLANAR TEST DATA
 0.055E6,  0.0690
 0.324E6,  0.2828
 0.758E6,  1.3862
 1.269E6,  3.0345
 1.779E6,  4.0621
```

The input file for the single-element simulation of the three experimental tests is shown in "Test fit of hyperelastic material data," Section A.11. The computational and experimental results for the various types of tests are compared in Figure 10–47, Figure 10–48, and Figure 10–49. The Abaqus and experimental results for the biaxial tension test match very well. The computational and experimental results for the uniaxial tension and planar tests match well at strains less than 100%. The hyperelastic material model created from these material test data is probably not suitable for use in general simulations where the strains may be larger than 100%. However, the model will be adequate for this simulation if the principal strains remain within the strain magnitudes where the data and the hyperelastic model fit well. If you find that the results are beyond these magnitudes or if you are asked to perform a different simulation, you will have to insist on getting better material data. Otherwise, you will not be able to have much confidence in your results.

Material properties: elastic properties for the steel

The steel is modeled with linear elastic properties only (E = 200 GPa, ν = 0.3) because the loads should not be large enough to cause inelastic deformations. Thus, the material option blocks for the steel are

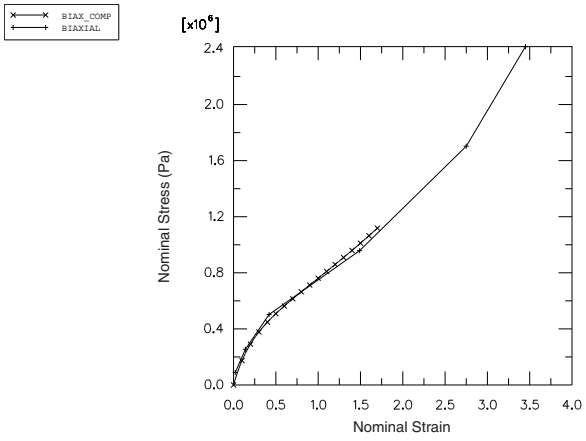


Figure 10–47 Comparison of experimental data (BIAXIAL) and Abaqus results (BIAX_COMP): biaxial tension.

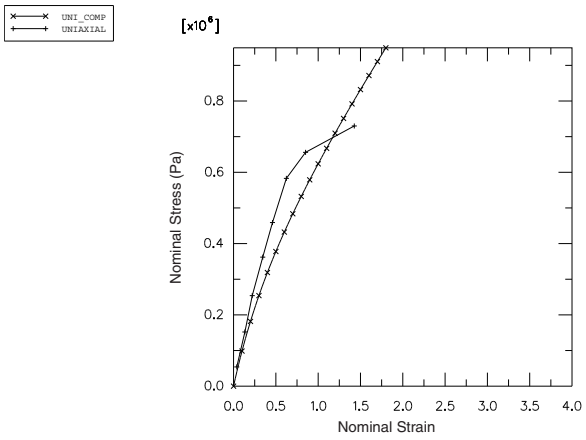


Figure 10–48 Comparison of experimental data (UNIAXIAL) and Abaqus results (UNI_COMP): uniaxial tension.

```
*MATERIAL, NAME=STEEL
*ELASTIC
2.0E11, 0.3
```

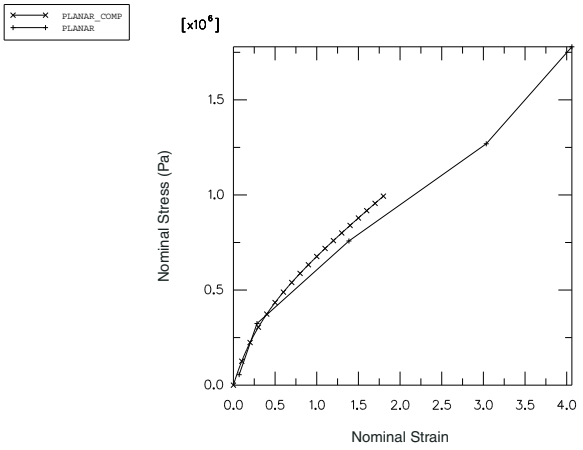


Figure 10–49 Comparison of experimental data (PLANAR) and Abaqus results (PLANAR_COMP): planar shear.

10.7.6 Reviewing the input—the history data

We now discuss the history data associated with this problem, including the time incrementation parameters, boundary conditions, loading, and output requests.

Including nonlinear geometry and specifying the initial increment size

When hyperelastic materials are used in a model, Abaqus assumes that it may undergo large deformations. But large deformations and other nonlinear geometric effects are included only if the NLGEOM parameter is set to YES on the ∗STEP option. Therefore, you must include it in this simulation or Abaqus will terminate the analysis with an input error. The ∗STEP option should look like

```
*STEP, NLGEOM=YES
```

The simulation will be a static analysis with a total step time of 1.0. Specify the initial time increment to be 1/100th of the total step time. The procedure option block should look like

```
*STATIC
.01, 1.0
```

Boundary conditions

Specify symmetry boundary conditions on the nodes lying on the symmetry plane. In this model the symmetry conditions prevent the nodes from moving in degree of freedom 2 (axially), as shown in Figure 10–50.

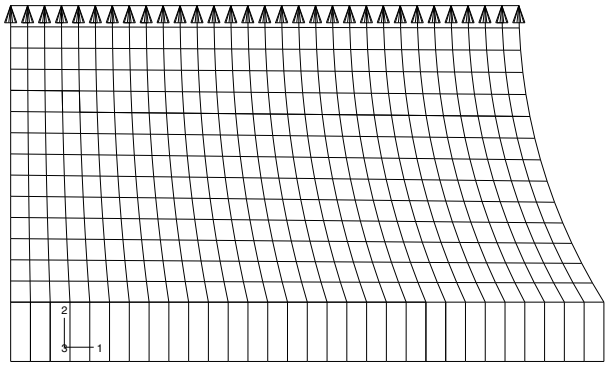


Figure 10–50 Boundary conditions on the rubber mount.

Symmetry conditions that constrain motion in the global 2-direction can be applied using the YSYMM type boundary condition, or you can simply constrain the 2-direction. In this case the ∗BOUNDARY option block has the following format:

```
*BOUNDARY
MIDDLE, 2, 2, 0.0
```

No boundary constraints are needed in the radial direction (global 1-direction) because the axisymmetric nature of the model does not allow the structure to move as a rigid body in the radial direction. Abaqus will allow nodes to move in the radial direction, even those initially on the axis of symmetry (i.e., those with a radial coordinate of 0.0), if no boundary conditions are applied to their radial displacements (degree of freedom 1). Since you want to let the mount deform radially in this analysis, do not apply any boundary conditions; again, Abaqus will prevent rigid body motions automatically.

Loading

The mount must carry a maximum axial load of 5.5 kN, spread uniformly over the steel plates. A distributed load is, therefore, applied to the bottom of the steel plate. The magnitude of the pressure is given by

$$p = 5500/\pi(0.06^2 - 0.01^2) \cong 0.50\text{MPa}$$

If you generated the pressure loading using a preprocessor, a ∗DLOAD option block with many data lines may be present in the input file.

```
*DLOAD
 1, P1, 0.50E6
```

```
 2, P1, 0.50E6
...
29, P1, 0.50E6
30, P1, 0.50E6
```

For the element and node numbering discussed here, the pressure is applied to face 1 of all the elements in element set **PLATE**. This allows us to use a much more compact format for the data lines of the *DLOAD option block.

```
*DLOAD
PLATE, P1, 0.50E6
```

Output requests

Write the preselected variables and nominal strains as field output to the output database file. In addition, write the displacement of one of the nodes on the bottom of the steel plate to the output database file so that the stiffness of the mount can be calculated. You will need to create a node set containing the node. The output option blocks in your model should be similar to the following:

```
*NSET, NSET=OUT
1,
*OUTPUT, FIELD, VARIABLE=PRESELECT
*ELEMENT OUTPUT
NE,
*OUTPUT, HISTORY
*NODE OUTPUT, NSET=OUT
U,
```

Ensure that the end of the step definition is clearly marked with an *END STEP option.

10.7.7 Running the analysis

Store your input options in a file called **mount.inp**. The input options for the model discussed in the above sections can be found in "Axisymmetric mount," Section A.10. Since the nonlinear nature of the simulation means that it may take some time to complete, use the following command to run the analysis in the background:

```
abaqus job=mount
```

When the job has completed, check the data file, **mount.dat**, for errors. If there are any, correct the input file and rerun the analysis. If necessary, compare your input with that shown in "Axisymmetric mount," Section A.10.

10.7.8 Results

We briefly review the results associated with the polynomial fit of the test data.

The hyperelastic material parameters

In this simulation you specified that the material is incompressible (D_1=0). The incompressibility is assumed since no volumetric test data were provided. To simulate compressible behavior, you must provide volumetric test data in addition to the other test data. You also specified that Abaqus should use a first-order, polynomial strain energy function. This form of the hyperelasticity model is known as the Mooney-Rivlin material model.

The hyperelastic material coefficients—C_{10}, C_{01}, and D_1—that Abaqus calculates from the material test data are given in the data file, **`mount.dat`**, provided that you used the *PREPRINT, MODEL=YES option in the model data section of the input file. The material test data are also written in the file so that you can ensure that Abaqus used the correct data, as shown below.

```
                          M A T E R I A L   D E S C R I P T I O N


 MATERIAL NAME: RUBBER

    HYPERELASTIC MATERIAL PROPERTIES


         UNIAXIAL TEST DATA

         NOMINAL STRAIN      NOMINAL STRESS(TEST)      NOMINAL STRESS(ABAQUS)
            3.8000E-02              5.4000E+04              3.9605E+04
            0.1338                  1.5200E+05              1.2803E+05
            0.2210                  2.5400E+05              1.9764E+05
            0.3450                  3.6200E+05              2.8404E+05
            0.4600                  4.5900E+05              3.5477E+05
            0.6242                  5.8300E+05              4.4505E+05
            0.8510                  6.5600E+05              5.5627E+05
             1.427                  7.3000E+05              8.0275E+05

    HYPERELASTIC MATERIAL PROPERTIES


         BIAXIAL TEST DATA

         NOMINAL STRAIN      NOMINAL STRESS(TEST)      NOMINAL STRESS(ABAQUS)
            2.0000E-02              8.9000E+04              4.1264E+04
            0.1400                  2.5500E+05              2.2551E+05
            0.4200                  5.0300E+05              4.6078E+05
             1.490                  9.5800E+05              1.0063E+06
             2.750                  1.7030E+06              1.7767E+06
             3.450                  2.4130E+06              2.3301E+06

    HYPERELASTIC MATERIAL PROPERTIES


         PLANAR TEST DATA
```

```
     NOMINAL STRAIN      NOMINAL STRESS(TEST)      NOMINAL STRESS(ABAQUS)
        6.9000E-02              5.5000E+04              9.0339E+04
        0.2828                  3.2400E+05              2.9189E+05
         1.386                  7.5800E+05              8.3431E+05
         3.034                  1.2690E+06              1.4500E+06
         4.062                  1.7790E+06              1.8235E+06


 HYPERELASTICITY - MOONEY-RIVLIN STRAIN ENERGY

                  D1                C10               C01

              0.00000000        176050.524        4332.63031
```

If there were any problems with the stability of the hyperelastic material model, warning messages would be given before the material parameters. The material model is stable at all strains with these material test data and this strain energy function. However, if you specified that a second-order (N=2), polynomial strain energy function be used, you would see the following warnings in the data file:

```
  *HYPERELASTIC, N=2, POLYNOMIAL, TEST DATA INPUT
***WARNING: UNSTABLE HYPERELASTIC MATERIAL
        FOR UNIAXIAL TENSION WITH A NOMINAL STRAIN LARGER THAN        6.9700
        FOR UNIAXIAL COMPRESSION WITH A NOMINAL STRAIN LESS THAN     -0.9795
        FOR BIAXIAL TENSION WITH A NOMINAL STRAIN LARGER THAN         5.9800
        FOR BIAXIAL COMPRESSION WITH A NOMINAL STRAIN LESS THAN      -0.6458
        FOR PLANE TENSION WITH A NOMINAL STRAIN LARGER THAN           7.0400
        FOR PLANE COMPRESSION WITH A NOMINAL STRAIN LESS THAN        -0.8756

        POLYNOMIAL STRAIN ENERGY FUNCTION WITH N =  2
             D1          C10          C01
             D2          C20          C11          C02
         0.0000E+00  0.1934E+06  -148.2
         0.0000E+00  -805.7       180.0       -3.967
```

If you only had the uniaxial test data available for this problem, you would find that the Mooney-Rivlin material model Abaqus creates would have unstable material behavior above certain strain magnitudes.

10.7.9 Postprocessing

Use Abaqus/Viewer to look at the analysis results by issuing the following command at the operating system prompt:

```
abaqus viewer odb=mount
```

Calculating the stiffness of the mount

Determine the stiffness of the mount by creating an *X*–*Y* plot of the displacement of the steel plate as a function of the applied load. You will first create a plot of the vertical displacement of the node on the steel plate for which you wrote data to the output database file. Data were written for the node in set **OUT** in this model.

To create a history curve of vertical displacement and swap the *X*- and *Y*-axes:

1. In the Results Tree, expand the **History Output** container underneath the output database named **`mount.odb`**.
2. Locate and select the vertical displacement U2 at the node in set **`OUT`**.
3. Click mouse button 3, and select **Save As** from the menu that appears to save the *X*–*Y* data.

   The **Save XY Data As** dialog box appears.
4. In the **Save XY Data As** dialog box, name the curve **`SWAPPED`** and select **swap(XY)** as the save operation; click **OK**.

   The plot of time-displacement appears in the viewport.

You now have a curve of time-displacement. What you need is a curve showing force-displacement. This is easy to create because in this simulation the force applied to the mount is directly proportional to the total time in the analysis. All you have to do to plot a force-displacement curve is multiply the curve **`SWAPPED`** by the magnitude of the load (5.5 kN).

To multiply a curve by a constant value:

1. In the Results Tree, double-click **XYData**.

   The **Create XY Data** dialog box appears.
2. Select **Operate on XY data**, and click **Continue**.

   The **Operate on XY Data** dialog box appears.
3. In the **XY Data** field, double-click **`SWAPPED`**.

   The expression **`"SWAPPED"`** appears in the text field at the top of the dialog box. Your cursor should be at the end of the text field.
4. Multiply the data object in the text field by the magnitude of the applied load by entering **`*5500`**.
5. Save the multiplied data object by clicking **Save As** at the bottom of the dialog box.

   The **Save XY Data As** dialog box appears.
6. In the **Name** text field, type **`FORCEDEF`**; and click **OK** to close the dialog box.
7. To view the force-displacement plot, click **Plot Expression** at the bottom of the **Operate on XY Data** dialog box.

You have now created a curve with the force-deflection characteristic of the mount (the axis labels do not reflect this since you did not change the actual variable plotted). To get the stiffness, you need to differentiate the curve **`FORCEDEF`**. You can do this by using the **`differentiate( )`** operator in the **Operate on XY Data** dialog box.

To obtain the stiffness:

1. In the **Operate on XY Data** dialog box, clear the current expression.
2. From the **Operators** listed, click **differentiate(X)**.
   `differentiate( )` appears in the text field at the top of the dialog box.
3. In the **XY Data** field, double-click `FORCEDEF`.
   The expression `differentiate( "FORCEDEF" )` appears in the text field.
4. Save the differentiated data object by clicking **Save As** at the bottom of the dialog box.
   The **Save XY Data As** dialog box appears.
5. In the **Name** text field, type `STIFF`; and click **OK** to close the dialog box.
6. To plot the stiffness-displacement curve, click **Plot Expression** at the bottom of the **Operate on XY Data** dialog box.
7. Click **Cancel** to close the dialog box.
8. Open the **Axis Options** dialog box, and switch to the **Title** tabbed page.
9. Customize the axis titles so they appear as shown in Figure 10–51.

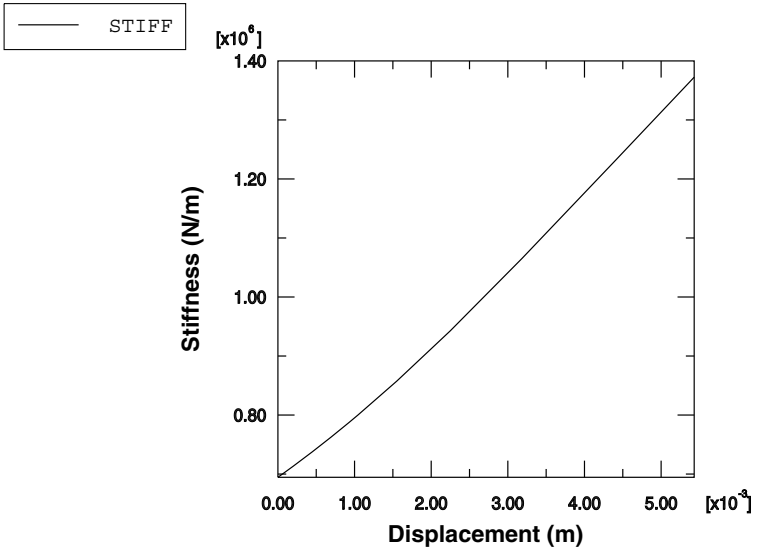


Figure 10–51 Stiffness characteristic of the mount.

10. Click **Dismiss** to close the **Axis Options** dialog box.

The stiffness of the mount increases by almost 100% as the mount deforms. This is a result of the nonlinear nature of the rubber and the change in shape of the mount as it deforms. Alternatively, you could have created the stiffness-displacement curve directly by combining all the operators above into one expression.

To define the stiffness curve directly:

1. In the Results Tree, double-click **XYData**.
   The **Create XY Data** dialog box appears.
2. Select **Operate on XY data**, and click **Continue**.
   The **Operate on XY Data** dialog box appears.
3. From the **Operators** listed, click **differentiate(X)**.
   `differentiate( )` appears in the text field at the top of the dialog box.
4. In the **XY Data** field, double-click `SWAPPED`.
   The expression `differentiate( "SWAPPED" )` appears in the text field.
5. Place the cursor in the text field directly after the `"SWAPPED"` data object, and type `*5500` to multiply the swapped data by the constant total force value.
   `differentiate( "SWAPPED"*5500 )` appears in the text field.
6. Save the differentiated data object by clicking **Save As** at the bottom of the dialog box.
   The **Save XY Data As** dialog box appears.
7. In the **Name** text field, type `STIFFNESS`; and click **OK** to close the dialog box.
8. Click **Cancel** to close the **Operate on XY Data** dialog box.
9. Customize the *X*- and *Y*-axis labels as they appear in Figure 10–51 if you have not already done so.
10. In the Results Tree, click mouse button 3 on `STIFFNESS` underneath the **XYData** container and select **Plot** from the menu that appears to view the plot in Figure 10–51 that shows the variation of the mount's axial stiffness as the mount deforms.

Model shape plots

You will begin by plotting the undeformed model shape of the mount.

To plot the undeformed model shape:

From the main menu bar, select **Plot→Undeformed Shape**; or use the tool in the Visualization module toolbox to plot the undeformed model shape (see Figure 10–52).

If the figure obscures the plot title, you can move the plot by clicking the tool and holding down mouse button 1 to pan the deformed shape to the desired location. Alternatively, you can turn the plot title off (**Viewport→Viewport Annotation Options**).

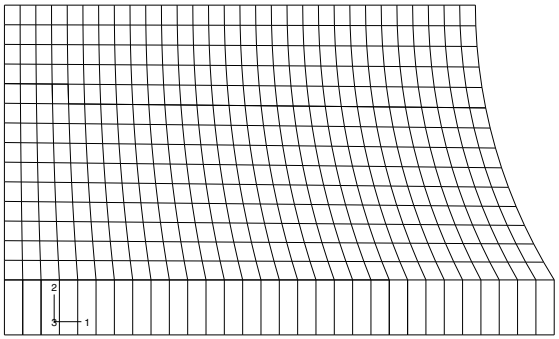


Figure 10–52 Undeformed model shape of the rubber mount.

In this figure the axisymmetric model is displayed as a planar, two-dimensional shape. You can producing a three-dimensional visual effect by sweeping the model through a specified angle. In addition, you can also mirror results about selected planes (such as symmetry planes) to render results on a full three-dimensional representation of the model. These are visualization aids only. Any numerical representation of the results, such as the contour legend, indicates only the portion of the model that was analyzed. Since in this problem the symmetry plane does not necessarily coincide with one of the global coordinate system planes, a local system will be defined to facilitate the mirroring operation.

To define a local coordinate system for postprocessing:

1. From the main menu bar, select **Tools→Coordinate System→Create**.
2. In the **Create Coordinate System** dialog box, enter **`rectangular`** as the name and click **Continue**.
3. Select the node at the top-left corner of the model as the origin, the node at the top-right corner as the point on the *X*-axis, and the node at the bottom-left corner as the point in the *X*–*Y* plane.

To mirror and sweep the cross-section:

1. From the main menu bar, select **View→ODB Display Options**.
2. In the **ODB Display Options** dialog box, click the **Mirror/Pattern** tab.
3. Select **rectangular** from the **Mirror CSYS** list.
4. Select **XZ** as the mirror plane.
5. Click **Apply**.
   The mirrored image appears.

6. Click the **Sweep/Extrude** tab.
7. Toggle on **Sweep elements** and set the sweep range from `0` to `270` degrees. Set the number of segments to `45`.
8. Click **OK**.

   The swept image appears. To more clearly distinguish between the rubber and the steel, color code the model based on section assignment.

You will now plot the deformed model shape of the mount. This will allow you to evaluate the quality of the deformed mesh and to assess the need for mesh refinement.

To plot the deformed model shape:

From the main menu bar, select **Plot→Deformed Shape**, or use the tool to plot the deformed model shape of the mount (see Figure 10–53).

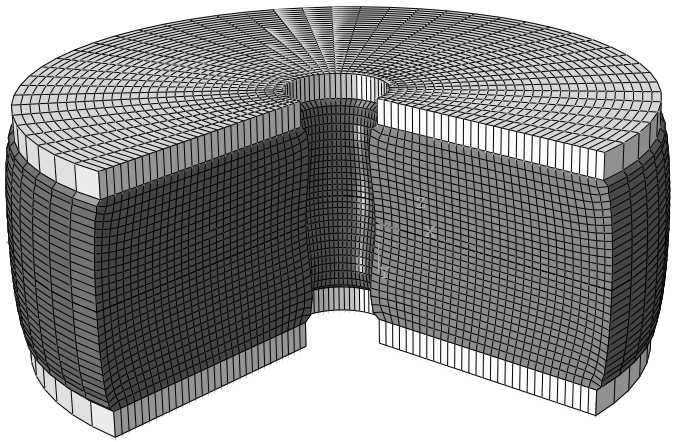


Figure 10–53 Deformed model shape of the rubber under an applied load of 5500 N (mirrored/swept image).

The plate has been pushed up, causing the rubber to bulge at the sides. Zoom in on the bottom left corner of the mesh using the tool from the **View Manipulation** toolbar. Click mouse button 1, and hold it down to define the first corner of the new view; move the mouse to create a box enclosing the viewing area that you want (Figure 10–54); and release the mouse button. Alternatively, you can zoom and pan the plot by selecting **View→Specify** from the main menu bar.

You should have a plot similar to the one shown in Figure 10–54 (in this and the images that follow, the sweep and mirroring operations applied earlier have been suppressed).

Some elements in this corner of the model are becoming badly distorted because the mesh design in this area was inadequate for the type of deformation that occurs there. Although the shape

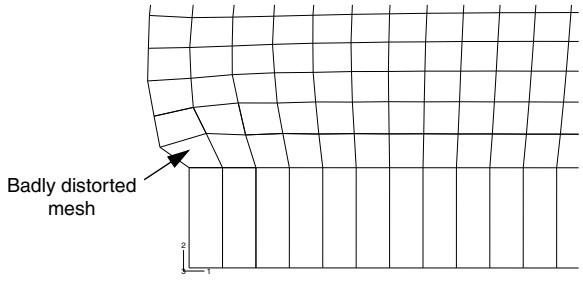


Figure 10–54 Distortion at the left-hand corner of the rubber mount model.

of the elements is fine at the start of the analysis, they become badly distorted as the rubber bulges outward, especially the element in the corner. If the loading were increased further, the element distortion may become so excessive that the analysis may abort. "Mesh design for large distortions," Section 10.8, discusses how to improve the mesh design for this problem.

The keystoning pattern exhibited by the distorted elements in the bottom right-hand corner of the model indicates that they are locking. A contour plot of the hydrostatic pressure stress in these elements (without averaging across elements sharing common nodes) shows rapid variation in the pressure stress between adjacent elements. This indicates that these elements are suffering from volumetric locking, which was discussed earlier in "Selecting elements for elastic-plastic problems," Section 10.3, in the context of plastic incompressibility. Volumetric locking arises in this problem from overconstraint. The steel is very stiff compared to the rubber. Thus, along the bond line the rubber elements cannot deform laterally. Since these elements must also satisfy incompressibility requirements, they are highly constrained and locking occurs. Analysis techniques that address volumetric locking are discussed in "Techniques for reducing volumetric locking," Section 10.9.

Contouring the maximum principal stress

Plot the maximum in-plane principal stress in the model. Follow the procedure given below to create a filled contour plot on the actual deformed shape of the mount with the plot title suppressed.

To contour the maximum principal stress:

1. By default, Abaqus/Viewer displays **S**, **Mises** as the primary field output variable. In the **Field Output** toolbar, select **Max. Principal** as the invariant.

   Abaqus/Viewer automatically changes the current plot state to display a contour plot of the maximum in-plane principal stresses on the deformed model shape.
2. Open the **Contour Plot Options** dialog box.
3. Drag the uniform contour intervals slider to **8**.
4. Click **OK** to view the contour plot and to close the dialog box.

Create a display group showing only the elements in the rubber mount.

5. In the Results Tree, expand the **Materials** container underneath the output database file named **`Mount.odb`**.
6. Click mouse button 3 on **RUBBER**, and select **Replace** from the menu that appears to replace the current display with the selected elements.
7. The viewport display changes and displays only the rubber mount elements, as shown in Figure 10–55.

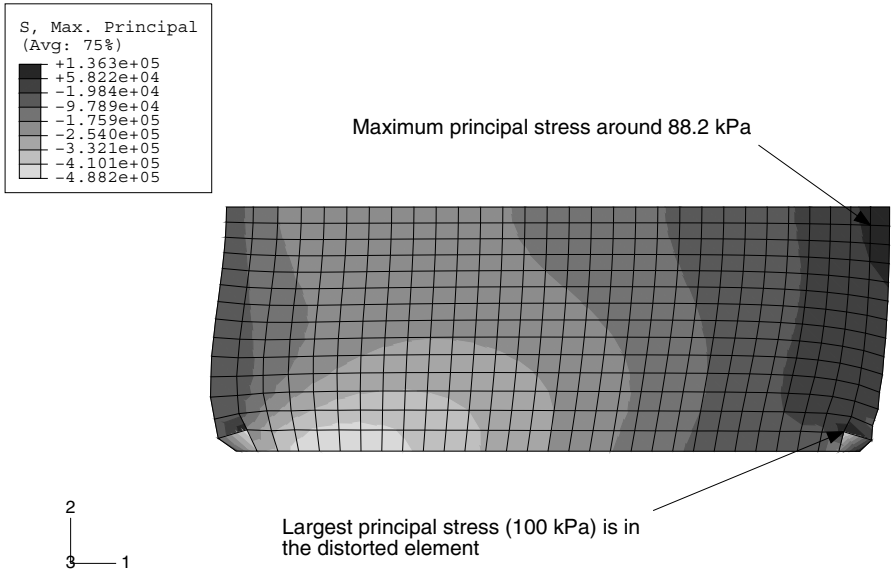


Figure 10–55 Contours of maximum principal stress in the rubber mount.

The maximum principal stress in the model, reported in the contour legend, is 136 kPa. Although the mesh in this model is fairly refined and, thus, the extrapolation error should be minimal, you may want to use the query tool to determine the more accurate integration point values of the maximum principal stress.

When you look at the integration point values, you will discover that the peak value of maximum principal stress occurs in one of the distorted elements in the bottom right-hand part of the model. This value is likely to be unreliable because of the levels of element distortion and

volumetric locking. If this value is ignored, there is an area near the plane of symmetry where the maximum principal stress is around 88.2 kPa.

The easiest way to check the range of the principal strains in the model is to display the maximum and minimum values in the contour legend.

To check the principal nominal strain magnitude:

1. From the main menu bar, select **Viewport→Viewport Annotation Options**.

   The **Viewport Annotation Options** dialog box appears.

2. Click the **Legend** tab, and toggle on **Show min/max values**.

3. Click **OK**.

   The maximum and minimum values appear at the bottom of the contour legend in the viewport.

4. In the **Field Output** toolbar, select **Primary** as the variable type if it is not already selected.

   Abaqus/Viewer automatically changes the current plot state to display a contour plot of the maximum in-plane principal stresses on the deformed model shape.

5. From the list of output variables, select **NE**.

6. From the list of invariants in the **Field Output** toolbar, select **Max. Principal** if it is not already selected.

   The contour plot changes to display values for maximum principal nominal strain. Note the value of the maximum principal nominal strain from the contour legend.

7. From the list of invariants, select **Min. Principal**.

   The contour plot changes to display values for minimum principal nominal strain. Note the value of the minimum principal nominal strain from the contour legend.

The maximum and minimum principal nominal strain values indicate that the maximum tensile nominal strain in the model is about 100% and the maximum compressive nominal strain is about 56%. Because the nominal strains in the model remained within the range where the Abaqus hyperelasticity model has a good fit to the material data, you can be fairly confident that the response predicted by the mount is reasonable from a material modeling viewpoint.

10.8 Mesh design for large distortions

We know that the element distortions in the corners of the rubber mount are undesirable. The results in these areas are unreliable; and if the load were increased, the analysis might fail. These problems can be corrected by using a better mesh design. The mesh shown in Figure 10–56 is an example of an alternate mesh design that might be used to reduce element distortion in the bottom left corner of the rubber model.

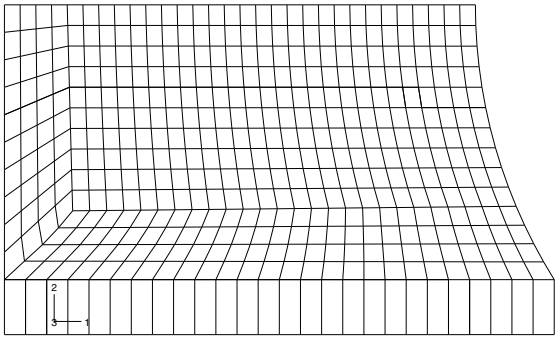


Figure 10–56 Modified mesh to minimize element distortions in the bottom left corner of the rubber model during the simulation.

The issues surrounding the mesh distortion in the opposite corner are addressed in "Techniques for reducing volumetric locking," Section 10.9. The elements in the bottom left-hand corner region are now much more distorted in the initial, undeformed configuration. However, as the analysis progresses and the elements deform, their shape actually improves. The deformed shape plot, shown in Figure 10–57, illustrates that the amount of element distortion in this region is reduced; however, the level of mesh distortion in the bottom right-hand corner of the rubber model is still significant.

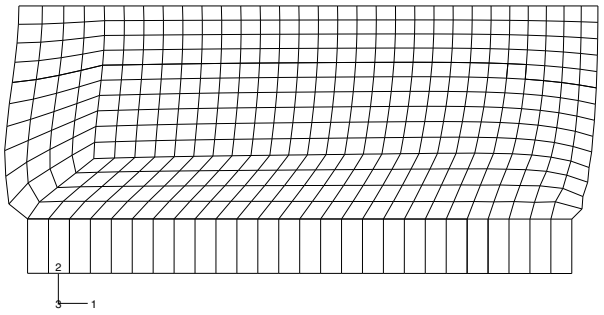


Figure 10–57 Deformed shape of the modified mesh.

The contours of maximum principal stress (Figure 10–58) show that the very localized stress in that corner has been reduced only slightly.

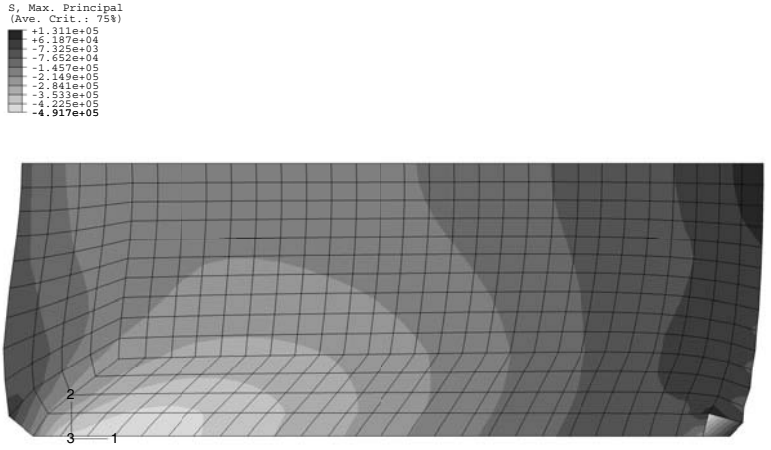


Figure 10–58 Contours of maximum principal stress in the modified mesh.

Mesh design for large-distortion problems is more difficult than it is for small-displacement problems. A mesh must be produced where the shape of the elements is reasonable throughout the analysis, not just at the start. You must estimate how the model will deform using experience, hand calculations, or the results from a coarse finite element model.

10.9 Techniques for reducing volumetric locking

A small amount of compressibility is introduced into the rubber material model in order to alleviate volumetric locking. Provided the amount of compressibility is small, the results obtained with a nearly incompressible material will be very similar to those obtained with an incompressible material.

Compressibility is introduced by setting the material constant D_1 to a nonzero value. The value is chosen so that the initial Poisson's ratio, ν_0, is close to 0.5. The equations given in "Hyperelastic behavior of rubberlike materials," Section 22.5.1 of the Abaqus Analysis User's Manual, can be used to relate D_1 and ν_0 in terms of μ_0 and K_0 (the initial shear and bulk moduli, respectively) for the polynomial form of the strain energy potential. For example, the hyperelastic material coefficients obtained earlier from the test data (see "The hyperelastic material parameters" in "Example: axisymmetric mount," Section 10.7) were given as $C_{10} = 176051$ and $C_{01} = 4332.63$; thus, setting $D_1 = 5.\text{E}{-7}$ yields $\nu_0 = 0.46$.

A model incorporating compressibility with additional mesh refinement (to reduce mesh distortion) is shown in Figure 10–59 (this mesh can be generated easily by changing the edge seeds in Abaqus/CAE or another preprocessor).

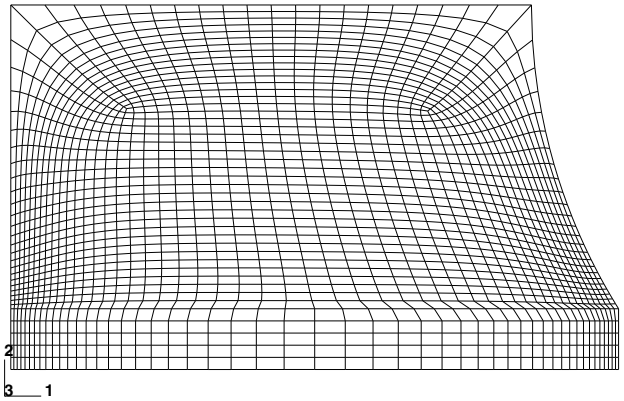


Figure 10–59 Modified mesh with refinement at both corners.

The deformed shape associated with this model is shown in Figure 10–60.

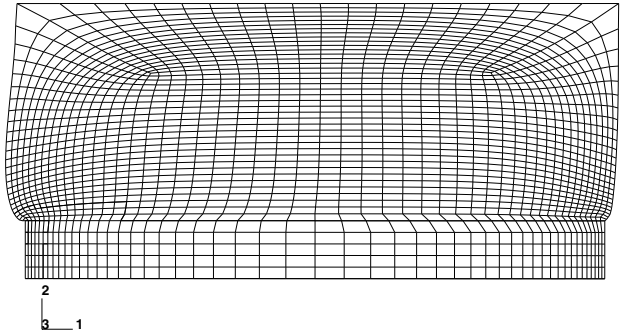


Figure 10–60 Deformed shape of the modified mesh.

It is clear from this figure that the mesh distortion has been reduced significantly in the critical regions of the rubber model. Examining contour plots of the pressure (without averaging across elements) reveals a smooth variation in pressure stress between elements. Thus, volumetric locking has been eliminated.

10.10 Related Abaqus examples

- "Pressurized rubber disc," Section 1.1.7 of the Abaqus Benchmarks Manual
- "Necking of a round tensile bar," Section 1.1.9 of the Abaqus Benchmarks Manual
- "Fitting of rubber test data," Section 3.1.4 of the Abaqus Benchmarks Manual

- "Uniformly loaded, elastic-plastic plate," Section 3.2.1 of the Abaqus Benchmarks Manual

10.11 Suggested reading

The following provides the interested user with additional references on material modeling.

General texts on materials

- Ashby, M. F., and D. R. H. Jones, *Engineering Materials*, Pergamon Press, 1980.
- Callister, W. D., *Materials Science & Engineering—An Introduction*, John Wiley, 1994.
- Pascoe, K. J., *An Introduction to the Properties of Engineering Materials*, Van Nostrand, 1978.

Plasticity

- SIMULIA, *Metal Inelasticity in Abaqus*.
- Lubliner, J., *Plasticity Theory*, Macmillan Publishing Co., 1990.
- Calladine, C. R., *Engineering Plasticity*, Pergamon Press, 1969.

Rubber elasticity

- SIMULIA, *Modeling Rubber and Viscoelasticity with Abaqus*.
- Gent, A., *Engineering with Rubber (How to Design Rubber Components)*, Hanser Publishers, 1992.

10.12 Summary

- Abaqus contains an extensive library to model the behavior of various engineering materials. It includes models for metal plasticity and rubber elasticity.
- The stress-strain data for the metal plasticity model must be defined in terms of true stress and true plastic strain.
- The metal plasticity model in Abaqus assumes incompressible plastic behavior.
- For efficiency Abaqus/Explicit *regularizes* user-defined material curves by fitting them with curves composed of equally spaced points.
- The hyperelastic material model in Abaqus/Standard allows true incompressibility. The hyperelastic material model in Abaqus/Explicit does not: the default Poisson's ratio for hyperelastic materials in Abaqus/Explicit is 0.475. Some analyses may require increasing Poisson's ratio to model incompressibility more accurately.
- Polynomial, Ogden, Arruda-Boyce, Marlow, van der Waals, Mooney-Rivlin, neo-Hookean, reduced polynomial, and Yeoh strain energy functions are available for rubber elasticity (hyperelasticity).

All models allow the material coefficients to be determined directly from experimental test data. The test data must be specified as nominal stress and nominal strain values.

- Stability warnings may indicate that a hyperelastic material model is unsuitable for the strain ranges you wish to analyze.
- The presence of symmetry can be used to reduce the size of a simulation since only part of the component needs to be modeled. The effect of the rest of the component is represented by applying appropriate boundary conditions.
- Mesh design for large-distortion problems is more difficult than for small-displacement problems. The elements in the mesh must not become too distorted at any stage of the analysis.
- Volumetric locking can be alleviated by permitting a small amount of compressibility. Care must be taken to ensure that the amount of compressibility introduced into the problem does not grossly affect the overall results.
- The *X*–*Y* plotting capabilities in Abaqus/Viewer allow data in curves to be manipulated to create new curves. Two curves or a curve and a constant can be added, subtracted, multiplied, or divided. Curves can also be differentiated, integrated, and combined.

11. Multiple Step Analysis

The general goal of an Abaqus simulation is to predict the response of a structure to applied loads. Recall that in a general sense the term *load* in Abaqus refers to anything that induces a change in the response of a structure from its initial state; for example, nonzero boundary conditions or applied displacements, point forces, pressures, fields, etc. In some cases loads are relatively simple, such as a single set of point loads on a structure. In other problems the loads applied to a structure can be very complex. For example, different loads may be applied to different portions of the model in a particular sequence over some period of time, or the magnitude of the loads may vary as a function of time. The term *load history* is used to refer to such complex loading of a model.

In Abaqus the user divides the complete load history of the simulation into a number of *steps*. Each step is a period of "time," specified by the user, for which Abaqus calculates the response of the model to a particular set of loads and boundary conditions. The user must specify the type of response, known as the analysis procedure, during each step and may change analysis procedures from step to step. For example, static dead loads, perhaps gravitational loads, could be applied to a structure in one step; and the dynamic response of the loaded structure to earthquake accelerations could be calculated in the next step. Both implicit and explicit analyses can contain multiple steps; however, implicit and explicit steps cannot be combined in the same analysis job. To combine a series of implicit and explicit steps, the results transfer (or import) capability can be used. This feature is discussed in "Transferring results between Abaqus/Explicit and Abaqus/Standard," Section 9.2.2 of the Abaqus Analysis User's Manual, and is not discussed further here.

Abaqus divides all of its analysis procedures into two main groups: linear perturbation and general. General analysis steps can be included in an Abaqus/Standard or an Abaqus/Explicit analysis; linear perturbation steps are available only in Abaqus/Standard. Loading conditions and "time" are defined differently for the two cases. Furthermore, the results from each type of procedure should be interpreted differently.

The response of the model during a general analysis procedure, known as a *general step*, may be either nonlinear or linear. In a step that uses a perturbation procedure, which is called a *perturbation step*, the response can only be linear. Abaqus/Standard treats such steps as a linear perturbation about the preloaded, predeformed state (known as the base state) created by any previous general steps; therefore, its capability for doing linear simulations is rather more general than that of a purely linear analysis program.

11.1 General analysis procedures

The starting point for each general step is the deformed state at the end of the last general step. Therefore, the state of the model evolves in a sequence of general steps as it responds to the loads defined in each

step. Any initial conditions (specified using the *INITIAL CONDITIONS option) define the starting point for the first general step in the simulation.

All general analysis procedures share the same concepts for applying loads and defining "time."

11.1.1 Time in general analysis steps

Abaqus has two measures of time in a simulation. The *total time* increases throughout all general steps and is the accumulation of the total step time from each general step. Each step also has its own time scale (known as the *step time*), which begins at zero for each step. Time varying loads and boundary conditions can be specified in terms of either time scale. The time scales for an analysis whose history is divided into three steps, each 100 seconds long, are shown in Figure 11–1.

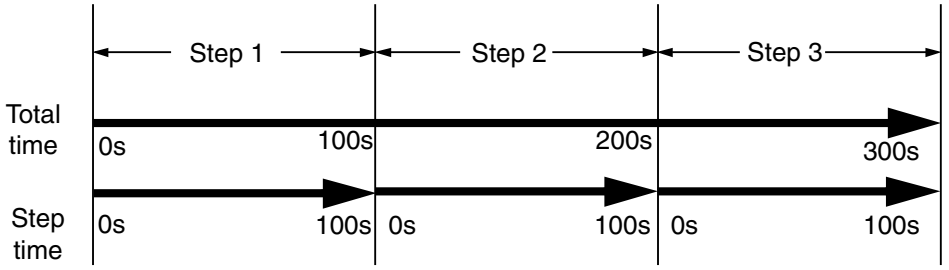


Figure 11–1 Step and total time for a simulation.

11.1.2 Specifying loads in general steps

In general steps the loads must be specified as total values, not incremental values. For example, if a concentrated load has a value of 1000 N in the first step and it is increased to 3000 N in the second general step, the magnitude given on the *CLOAD option in the two steps should be 1000 N and 3000 N, not 1000 N and 2000 N.

Modifying loads from step to step

Applying a load in Abaqus requires more than just providing its magnitude and direction. You must also specify how these new loads interact with the existing loads and boundary conditions of the same type that were defined in previous general steps. All the loading and boundary condition options—such as *BOUNDARY, *CLOAD, and *DLOAD—use the OP parameter to indicate how the loads they define interact with the existing loads of that type. The parameter can be set to OP=MOD or OP=NEW. Abaqus assumes OP=MOD if no value is provided for the OP parameter.

Using OP=MOD causes the loads defined in the current general step to modify the same types of loads already applied to the model in previous general steps. Any load that is not specifically modified in the current step continues to follow its associated amplitude definition, provided the amplitude curve is defined in terms of total time; otherwise, the load is maintained at the magnitude it had at the end of the last general step. For example, consider a cantilever beam modeled with two B22 elements (see Figure 11–2) with concentrated loads of 1000 N applied to nodes 3 and 5 in the first general step.

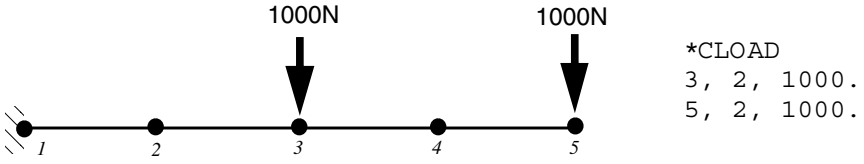


Figure 11–2 Loads applied to a beam in the first general step (Step 1).

In the next general step (Step 2) loads of 2000 N are specified on nodes 4 and 5 using the OP=MOD parameter. Thus, these loads modify those applied in Step 1. The loading applied to the model at the end of Step 2 is shown in Figure 11–3.

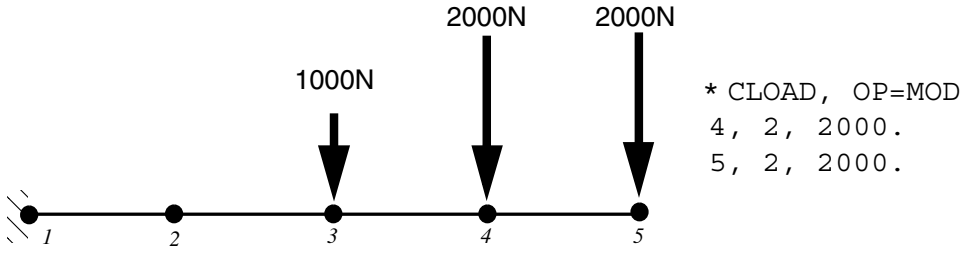


Figure 11–3 Loads applied in Step 2 with OP=MOD.

Using OP=NEW causes Abaqus to remove all existing loads of that type and only apply the loads specified in this current step to the model. If OP=NEW is specified on the ∗CLOAD option in Step 2, the loading on our example beam is shown in Figure 11–4.

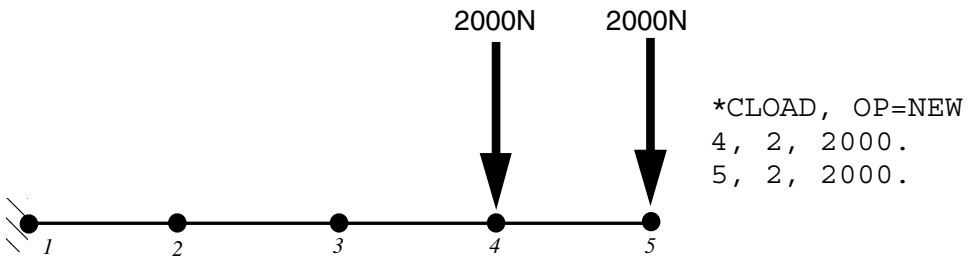


Figure 11–4 Loads applied in Step 2 with OP=NEW.

Be very careful when you use the OP=NEW parameter on the ∗BOUNDARY option to remove a boundary constraint from your model. All boundary constraints are removed from the model, not just the one you want removed; therefore, you must respecify all the boundary conditions that should remain active in the model. Remember that the boundary conditions that you specify must provide enough constraints to prevent rigid body motions in all components of your model. Failure to do so will cause Abaqus to issue numerical singularity warnings and leads to excessive displacements.

11.2 Linear perturbation analysis

Linear perturbation analysis steps are available only in Abaqus/Standard.

The starting point for a linear perturbation step is called the base state of the model. If the first step in a simulation is a linear perturbation step, the base state is the state of the model specified using the ∗INITIAL CONDITIONS option. Otherwise, the base state is the state of the simulation at the end of the last general step prior to the linear perturbation step. Although the response of the structure during the perturbation step is by definition linear, the model may have a nonlinear response in previous general steps. For models with a nonlinear response in the prior general steps, Abaqus/Standard uses the current elastic modulus as the linear stiffness for perturbation procedures. This modulus is the initial elastic modulus for elastic-plastic materials and the tangent modulus for hyperelastic materials (see Figure 11–5); the moduli used for other material models are described in “General and linear perturbation procedures,” Section 6.1.3 of the Abaqus Analysis User’s Manual.

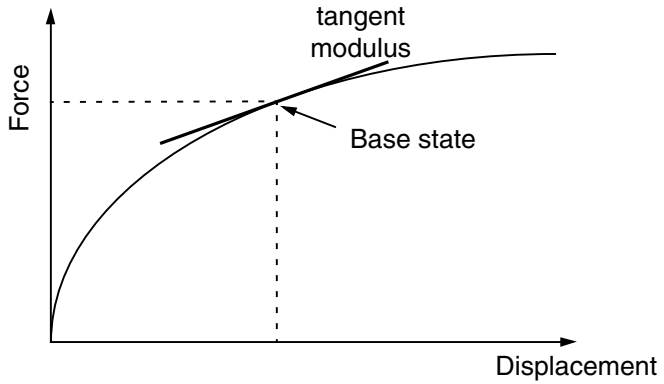


Figure 11–5 For hyperelastic materials the tangent modulus is used as the stiffness in linear perturbation steps that occur after general, nonlinear steps.

The loads in the perturbation step should be sufficiently small that the model's response would not deviate much from that predicted with the tangent modulus. If the simulation includes contact, the contact state between two surfaces does not change during a perturbation step: points that were closed in the base state remain closed, and points that were open remain open.

11.2.1 Time in linear perturbation steps

If another general step follows a perturbation step, Abaqus/Standard uses the state of the model at the end of the last general step as its starting point, not the state of the model at the end of the perturbation step. Thus, the response from a linear perturbation step has no permanent effect on the simulation. Therefore, Abaqus/Standard does not include the step time of linear perturbation steps in the total time for the analysis. In fact, what Abaqus/Standard actually does is to define the step time of a perturbation step to be very small (10^{-36}) so that it has no effect when it is added to the total accumulated time. The exception to this rule is the *MODAL DYNAMIC procedure.

11.2.2 Specifying loads in linear perturbation steps

Loads and prescribed boundary conditions given in linear perturbation steps are always local to that step. The load magnitudes (including the magnitudes of prescribed boundary conditions) given in a linear perturbation step are always the perturbation (increment) of the load, not the total magnitude. Likewise, the value of any solution variable is output as the perturbation value only—the value of the variable in the base state is not included.

As an example of a simple load history that includes a mixture of general and perturbation steps, consider the bow and arrow shown in Figure 11–6.

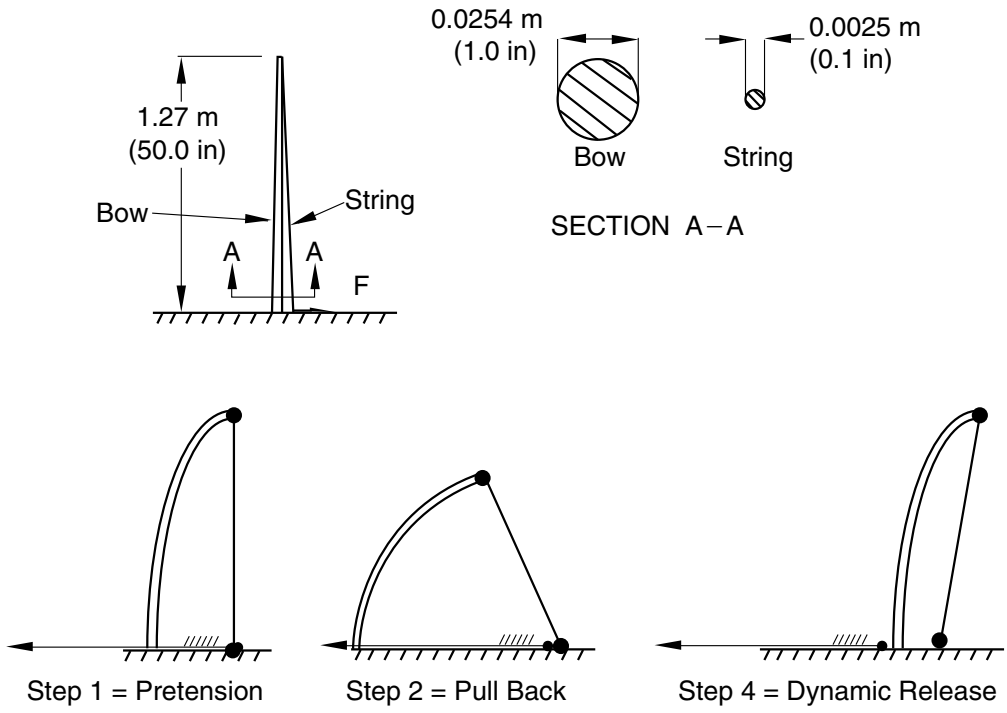


Figure 11–6 Simple bow and arrow.

Step 1 might be to string the bow—to pretension the bowstring. Step 2 would then follow this by pulling back the string with an arrow, thus storing more strain energy in the system. Step 3 might then be a linear perturbation analysis step: an eigenvalue frequency analysis to investigate the natural frequencies of the loaded system. Such a step might also have been included between Steps 1 and 2, to find the natural frequencies of the bow and string just after the string is pretensioned but before it is pulled back to shoot. Step 4 is then a nonlinear dynamic analysis, in which the bowstring is released, so that the strain energy that was stored in the system by pulling back the bowstring in Step 2 imparts kinetic energy to the arrow and causes it to leave the bow. This step thus continues to develop the nonlinear response of the system, but now with dynamic effects included.

In this case it is obvious that each nonlinear general analysis step must use the state at the end of the previous nonlinear general analysis step as its initial condition. For example, the dynamic part of the history has no loading—the dynamic response is caused by the release of some of the strain energy stored in the static steps. This effect introduces a natural order dependency in the model: nonlinear general analysis steps follow one another, in the order in which the events they define occur, with linear

perturbation analysis steps inserted at the appropriate times in this sequence to investigate the linear behavior of the system at those times.

A more complex load history is illustrated in Figure 11–7, which shows a schematic representation of the steps in the manufacture and use of a stainless steel sink. The sink is formed from sheet steel using a punch, a die, and a blank holder. This forming simulation will consist of a number of general steps. Typically, the first step may involve the application of blank holder pressure, and the punching operation will be simulated in the second step. The third step will involve the removal of the tooling, allowing the sink to spring back into its final shape. Each of these steps is a general step since together they model a sequential load history, where the starting condition for each step is the ending condition from the previous step. These steps obviously include many nonlinear effects (plasticity, contact, large deformations). At the end of the third step, the sink will contain residual stresses and inelastic strains caused by the forming process. Its thickness will also vary as a direct result of the manufacturing process.

The sink is then installed: boundary conditions would be applied around the edge of the sink where it is attached to the worktop. The response of the sink to a number of different loading conditions may be of interest and has to be simulated. For example, a simulation may need to be performed to ensure that the sink does not break if someone stands on it. Step 4 would, therefore, be a linear perturbation step analyzing the static response of the sink to a local pressure load. Remember that the results from Step 4 will be perturbations from the state of the sink after the forming process; do not be surprised, for example, if the displacement of the center of the sink in this step is say only 2 mm, but you know that the sink deformed much more than that since the start of the forming simulation. This hypothetical 2 mm deflection is just the additional deformation from the sink's final configuration after the forming process (i.e., the end of Step 3) caused by the weight of the person. The total deflection, measured from the undeformed sheet's configuration, is the sum of this 2 mm and the deflection at the end of Step 3.

The sink may also be fitted with a waste disposal unit, so its steady-state dynamic response to a harmonic load at certain frequencies must be simulated. Step 5 would, therefore, be a second linear perturbation step using the *STEADY STATE DYNAMICS, DIRECT procedure with a load applied at the point of attachment of the disposal unit. The base state for this step is the state at the end of the previous general step—that is, at the end of the forming process (Step 3). The response in the previous perturbation step (Step 4) is ignored. The two perturbation steps are, therefore, separate, independent simulations of the sink's response to loads applied to the base state of the model.

If another general step is included in the analysis, the condition of the structure at the start of the step is that at the end of the previous general step (Step 3). Step 6 could, therefore, be a general step with loads modeling the sink being filled with water. The response in this step may be linear or nonlinear. Following this general step, Step 7 could be a simulation repeating the analysis performed in Step 4. However, in this case the base state (the state of the structure at the end of the previous general step) is the state of the model at the end of Step 6. Therefore, the response will be that of a full sink, rather than an empty one. Performing another steady-state dynamics simulation would produce inaccurate results because the mass of the water, which would change the response considerably, would not be considered in the analysis.

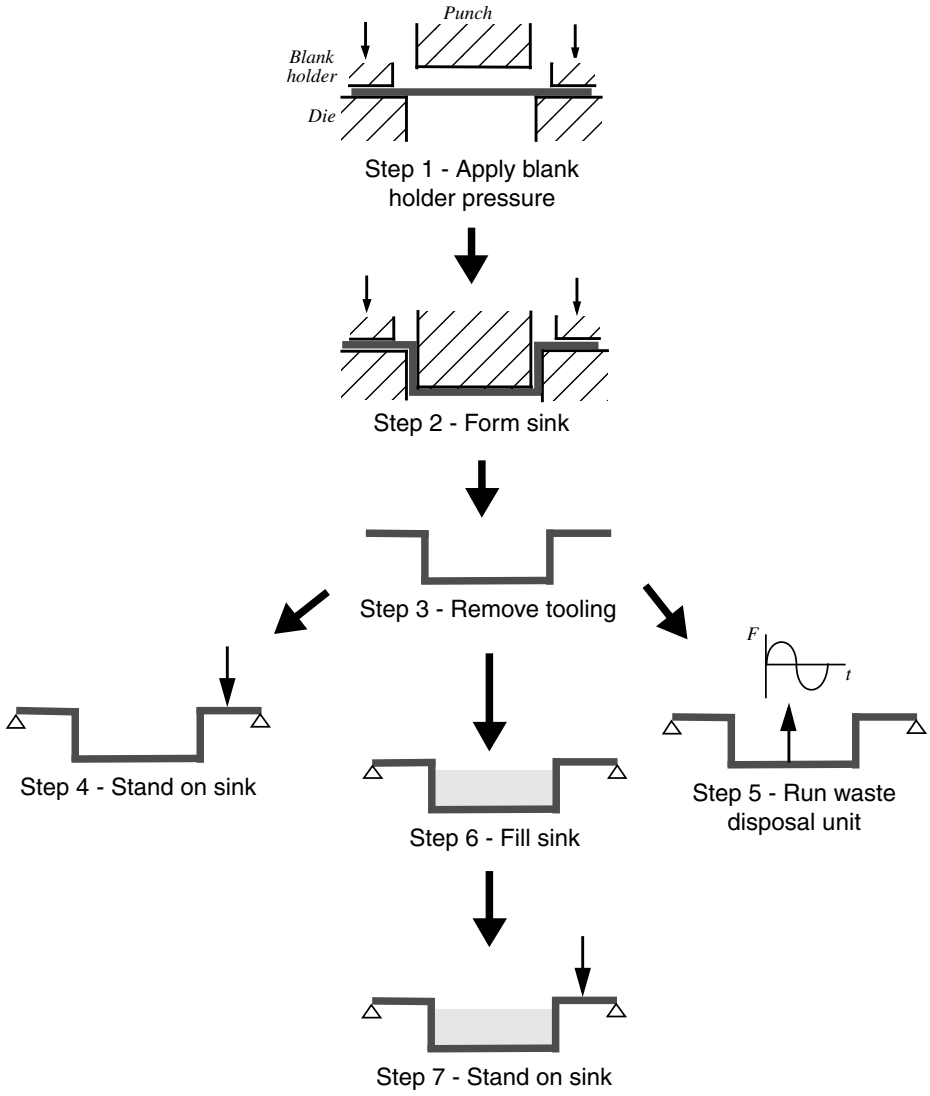


Figure 11–7 Steps in the manufacture and use of a sink.

The following procedures in Abaqus/Standard are always linear perturbation steps:

- *BUCKLE,
- *FREQUENCY,
- *MODAL DYNAMIC,
- *RANDOM RESPONSE,
- *RESPONSE SPECTRUM, and
- *STEADY STATE DYNAMICS.

The *STATIC procedure can be either a general or linear perturbation procedure. Include the PERTURBATION parameter on the *STEP option to make a static step a linear perturbation procedure.

11.3 Example: vibration of a piping system

In this example you will study the vibrational frequencies of a 5 m long section of a piping system. The pipe is made of steel and has an outer diameter of 18 cm and a 2 cm wall thickness (see Figure 11–8).

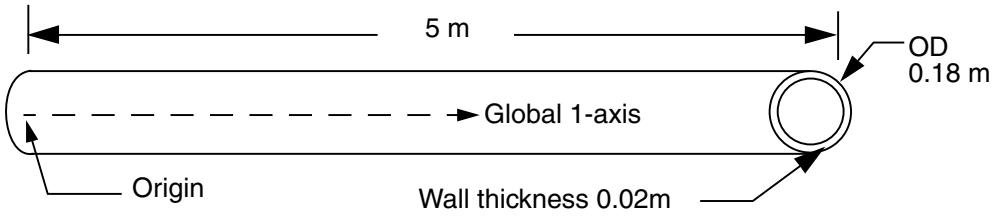


Figure 11–8 Portion of piping system being analyzed.

It is clamped firmly at one end and can move only axially at the other end. This 5 m portion of the piping system may be subjected to harmonic loading at frequencies up to 50 Hz. The lowest vibrational mode of the unloaded structure is 40.1 Hz, but this value does not consider how the loading applied to the piping structure may affect its response. To ensure that the section does not resonate, you have been asked to determine the magnitude of the in-service load that is required so that its lowest vibrational mode is higher than 50 Hz. You are told that the section of pipe will be subjected to axial tension when in service. Start by considering a load magnitude of 4 MN.

The lowest vibrational mode of the pipe will be a sine wave deformation in any direction transverse to the pipe axis because of the symmetry of the structure's cross-section. You will use three-dimensional beam elements to model the pipe section.

The analysis requires a natural frequency extraction. Thus, you will use Abaqus/Standard as your analysis product.

11.3.1 Coordinate system

The default, global coordinate system is used. Place the origin at the left end of the pipe section, and make the axis of the pipe and the global 1-axis coincident, as shown in Figure 11–8.

11.3.2 Mesh design

Model the pipe section with a uniformly spaced mesh of 30 second-order, pipe elements (PIPE32). The node and element numbers of the model used in this discussion are shown in Figure 11–9.

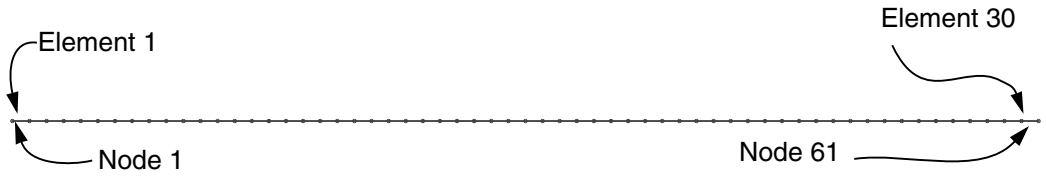


Figure 11–9 Node and element numbers (both increase by 1 from left to right).

11.3.3 Preprocessing—creating the model

You can create the mesh for this example using your preprocessor, or if you prefer, you can use the Abaqus mesh generation options shown in "Vibration of a piping system," Section A.12. If you wish to create the entire model using Abaqus/CAE please refer to "Example: vibration of a piping system," Section 11.3 of Getting Started with Abaqus: Interactive Edition.

11.3.4 Reviewing the input file—the model data

The steps that follow assumes that you have access to the full input file for this example. This input file, **`pipe-2.inp`**, is provided in "Vibration of a piping system," Section A.12, in the online HTML version of this manual. Instructions on how to fetch and run the script are given in Appendix A, "Example Files."

The model definition—including the model description, node and element definitions, section properties, and material properties—is discussed next.

Model description

The ∗HEADING option should include a suitable title in the data lines. In the sample input file, this option looks like the following:

```
*HEADING
Analysis of a 5 meter long pipe under tensile load
Pipe has OD of 180 mm and ID of 140 mm
S.I. Units
```

Nodal coordinates and element connectivity

Check that the correct element type (PIPE32) has been used and that the element set names are suitably descriptive.

```
*ELEMENT, TYPE=PIPE32, ELSET=PIPE
```

Create node sets containing the nodes at either end of the pipe section. The following option blocks create the node sets for the model shown in Figure 11–9:

```
*NSET, NSET=LEFT
1
*NSET, NSET=RIGHT
61
```

Beam properties

The ∗BEAM SECTION, SECTION=PIPE option will be used with the PIPE32 elements. The outer radius (90 mm) and the wall thickness (20 mm) are needed to define this beam section type geometrically. It is easier to define the orientation of the beam section geometry for this model than it was for the cargo crane model in the earlier chapters because the pipe section is symmetric. Define the approximate $\mathbf{n_1}$-direction as the vector (0., 0., –1.0). In this model the actual n_1-vector will coincide with this approximate vector.

```
*BEAM SECTION, ELSET=PIPE, MATERIAL=STEEL, SECTION=PIPE
0.09, 0.02
0.0, 0.0, -1.0
```

Material data

The option blocks defining the material behavior of the steel pipe in your model are included in the following lines:

```
*MATERIAL, NAME=STEEL
*ELASTIC
200.E9, 0.3
```

You must define the density of the steel material (7800 kg/m^3) because eigenmodes and eigenfrequencies are being extracted in this simulation and a mass matrix is needed for this procedure. Therefore, the following option block must follow the ∗ELASTIC option block:

```
*DENSITY
7800.,
```

11.3.5 Reviewing the input file—the history data

In this simulation you need to investigate the eigenmodes and eigenfrequencies of the steel pipe section when a 4 MN tensile load is applied. Therefore, the load history data will be split into two steps:

Step 1. General step:	Apply a 4 MN tensile force.
Step 2. Linear perturbation step:	Calculate modes and frequencies.

The actual magnitude of time in these steps will have no effect on the results; unless the model includes damping or rate-dependent material properties, "time" has no physical meaning in a static analysis procedure. Therefore, use a step time of 1.0 in the general analysis steps.

Step 1 – Apply a 4 MN tensile force

The options necessary to define the first analysis step—including the procedure definition, boundary conditions, loading, and output requests—are reviewed.

Step and analysis procedure definition

The first step is a general static step that includes the effect of geometric nonlinearity. Specify an initial increment size that is 1/10 the total step time, causing Abaqus to apply 10% of the load in the first increment. The following option blocks define the analysis procedure, and they include a meaningful description of the step to make reviewing the load history much easier:

```
*STEP, NLGEOM=YES
Apply axial tensile load of 4.0 MN
*STATIC
0.1, 1.0
```

Boundary conditions

The pipe section is clamped at its left end (node 1 in the model shown in Figure 11–9). It is also clamped at the other end; however, the axial force must be applied at this end, so only degrees of freedom 2 to 6 are constrained.

```
*BOUNDARY
LEFT, 1, 6
RIGHT, 2, 6
```

Tensile loading

Apply a 4 MN tensile force to the right end of the pipe section such that it deforms in the positive axial (global 1) direction. Forces are applied, by default, in the global coordinate system. Therefore, the ∗CLOAD option block looks like

```
*CLOAD
RIGHT, 1, 4.0E6
```

In this case the load is applied directly to the node set defined earlier. Use the node set name from your model or the node number in the *CLOAD option in your input file.

Output requests

Write data to the restart file every 10 increments. In addition, write the preselected field data every 10 increments as well as the stress components and stress invariants for element 25 as history data to the output database file. The following option blocks define these output requests:

```
*ELSET, ELSET=ELEMENT25
25
*RESTART, WRITE, FREQUENCY=10
*OUTPUT, FIELD, FREQUENCY=10, VARIABLE=PRESELECT
*OUTPUT, HISTORY
*ELEMENT OUTPUT, ELSET=ELEMENT25
S, SINV
```

End the step with the *END STEP option.

Step 2 – Extract modes and frequencies

The second step extracts the natural frequencies of the extended pipe. The required options are discussed below.

Step and analysis procedure definition

In the second step you need to calculate the eigenmodes and eigenfrequencies of the pipe in its loaded state. The eigenfrequency extraction procedure (*FREQUENCY option) used in this step is a linear perturbation procedure. Although only the first (lowest) eigenmode is of interest, extract the first eight eigenmodes for the model. Specify this number on the data line of the *FREQUENCY option block. The option blocks to define the analysis procedure should look similar to the following:

```
*STEP, PERTURBATION
Extract modes and frequencies
*FREQUENCY
8,
```

Loads

You require the natural frequencies of the extended pipe section. This does not involve the application of any perturbation loads, and the fixed boundary conditions are carried over from the previous general step. Therefore, you do not need to specify any loads or boundary conditions in this step.

Output requests

Any output requests required in a linear perturbation step must be redefined since the requests from the previous general step do not carry over. You want data to be written to the restart and output database files. The following option blocks define these requests:

```
*RESTART, WRITE
*OUTPUT, FIELD, VARIABLE=PRESELECT
```

Again, mark the termination of the step definition with the *END STEP option.

11.3.6 Running the analysis

Store the input option blocks in a file called **pipe.inp**. Run the analysis in the background using the command

```
abaqus job=pipe
```

11.3.7 Status file

Check the status file as the job is running. When the analysis completes, the contents of the status file will look similar to

```
SUMMARY OF JOB INFORMATION:
STEP   INC ATT SEVERE EQUIL TOTAL  TOTAL      STEP       INC OF       DOF    IF
               DISCON ITERS ITERS  TIME/    TIME/LPF    TIME/LPF    MONITOR RIKS
               ITERS                FREQ
  1     1   1     0     1     1  0.100       0.100       0.1000
  1     2   1     0     1     1  0.200       0.200       0.1000
  1     3   1     0     1     1  0.350       0.350       0.1500
  1     4   1     0     1     1  0.575       0.575       0.2250
  1     5   1     0     1     1  0.913       0.913       0.3375
  1     6   1     0     1     1  1.00        1.00        0.08750
  2     1   1     0     4     0  1.00        1.00e-36    1.000e-36
```

Both steps are shown, and the time associated with the linear perturbation step (Step 2) is very small: the *FREQUENCY procedure, or any linear perturbation procedure, does not contribute to the general loading history of the model.

11.3.8 Postprocessing

Run Abaqus/Viewer using the command

```
abaqus viewer odb=pipe
```

Deformed shapes from the linear perturbation steps

When Abaqus/Viewer starts, it automatically uses the last available frame on the output database file. The results from the second step of this simulation are the natural mode shapes of the pipe and the corresponding natural frequencies. Plot the first mode shape.

To plot the first mode shape:

1. From the main menu bar, select **Result→Step/Frame**.
   The **Step/Frame** dialog box appears.
2. Select step `Step-2` and frame `Mode 1`.
3. Click **OK**.
4. From the main menu bar, select **Plot→Deformed Shape**.
5. Click the tool in the toolbox to allow multiple plot states in the viewport; then click the tool or select **Plot→Undeformed Shape** to add the undeformed shape plot to the existing deformed plot in the viewport.
6. Include node symbols on both plots (the superimpose options control the appearance of the undeformed shape when multiple plot states are displayed). Change the color of the node symbols to green and the symbol shape to a solid circle.
7. Click the auto-fit tool so that the entire plot is rescaled to fit in the viewport.

The default view is isometric. Try rotating the model to find a better view of the first eigenmode, similar to that shown in Figure 11–10.

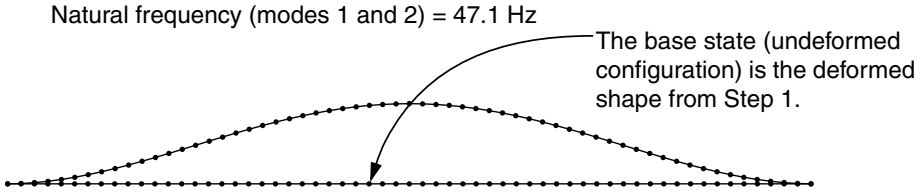


Figure 11–10 First and second eigenmode shapes of the pipe section under the tensile load (the modes lie in planes orthogonal to each other).

Since this is a linear perturbation step, the undeformed shape is the shape of the structure in the base state. This makes it easy to see the motion relative to the base state. Use the **Frame Selector** to plot the other mode shapes. You will discover that this model has many repeated eigenmodes. This is a result of the symmetric nature of the pipe's cross-section, which yields two eigenmodes

for each natural frequency, corresponding to the 1–2 and 1–3 planes. The second eigenmode shape is shown in Figure 11–10. Some of the higher vibrational mode shapes are shown in Figure 11–11.

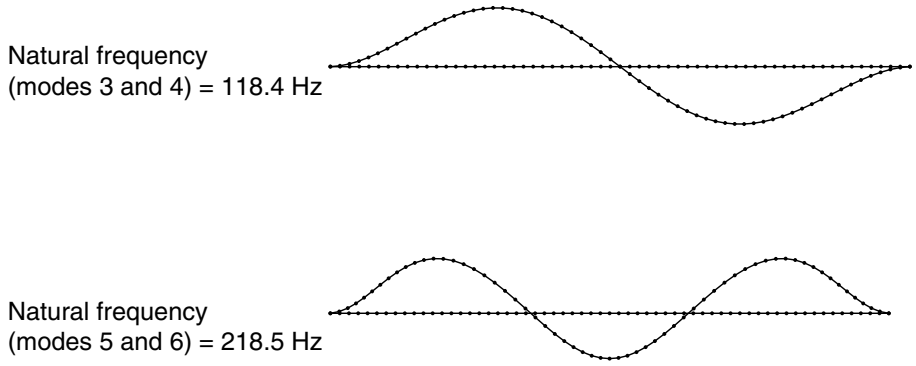


Figure 11–11 Shapes of eigenmodes 3 through 6; corresponding mode shapes lie in planes orthogonal to each other.

The natural frequency associated with each eigenmode is reported in the plot title. The lowest natural frequency of the pipe section when the 4 MN tensile load is applied is 47.1 Hz. The tensile loading has increased the stiffness of the pipe and, thus, increased the vibrational frequencies of the pipe section. This lowest natural frequency is within the frequency range of the harmonic loads; therefore, resonance of the pipe may be a problem when it is used with this loading.

You, therefore, need to continue the simulation and apply additional tensile load to the pipe section until you find the magnitude that raises the natural frequency of the pipe section to an acceptable level. Rather than repeating the analysis and increasing the applied axial load, you can use the restart capability in Abaqus to continue the load history of a prior simulation in a new analysis.

11.4 Restart analysis

Multistep simulations need not be defined in a single job. Indeed, it is usually desirable to run a complex simulation in stages. This allows you to examine the results and confirm that the analysis is performing as expected before continuing with the next stage. The Abaqus restart analysis capability allows a simulation to be restarted and the model's response to additional load history to be calculated.

The restart analysis capability is discussed in detail in "Restarting an analysis," Section 9.1.1 of the Abaqus Analysis User's Manual.

11.4.1 The restart and state files

The Abaqus/Standard restart (**.res**) file and the Abaqus/Explicit state (**.abq**) file contain the information necessary to continue a previous analysis. In Abaqus/Explicit the package (**.pac**) file and the selected results (**.sel**) file are also used for restarting an analysis and must be saved upon completion of the first job. In addition, both products require the output database (**.odb**) file. Restart files can become very large for large models; when restart data are requested, they are written for every increment or interval by default. Thus, it is important to control the frequency at which restart data are written. Sometimes it is useful to allow data to be overwritten on the restart file during a step. This means that at the end of the analysis there is only one set of restart data for each step, corresponding to the state of the model at the end of each step. However, if the analysis is interrupted for some reason, such as a computer malfunction, the analysis can be continued from the point where restart data were last written.

11.4.2 Writing a restart file

The *RESTART option controls the writing of the restart file. While the option can appear anywhere in the input file, it normally appears as part of a step definition. As with other output request options, the values defined in a *RESTART option apply during the current step and any subsequent general steps, until the option is modified in a later step. The option

```
*RESTART, WRITE, FREQUENCY=<n>
```

writes data to the restart file every *n*th increment. Restart data are also written at the end of each step, regardless of whether the last increment is divisible by *n*. If the FREQUENCY parameter is omitted, data are written every increment.

Restart files can become very large for large models, so it is often useful to include the OVERLAY parameter to control the size of the restart file. This parameter allows data to be overwritten on the restart file during a step.

11.4.3 Reading a restart file

When restarting a simulation from the end of a previous analysis, use the READ parameter on the *RESTART option. You can also use the STEP and INC parameters to specify the particular point in the simulation's load history from which to restart the analysis. When performing a restart simulation, the *RESTART option should appear immediately after the *HEADING option. No model data need appear in this restart input file since the model data for the analysis will be read from the restart file. Only node set definitions, element set definitions, amplitude definitions, and additional history data can be modified in the restart input file.

Continuing an interrupted run

The new analysis continues directly from the specified step and increment of the previous analysis. If the given step and increment do not correspond to the end of the previous analysis, Abaqus will simulate all of the remaining previously defined load history data before trying to simulate any new load history data provided in the input file. Therefore, if an analysis was interrupted by a computer malfunction, the following input file would complete the analysis as originally defined:

```
*HEADING
Restart of interrupted run
*RESTART, READ, STEP=<step>, INC=<increment>
```

Continuing with additional steps

If the previous analysis completed successfully and, having viewed the results, you want to add additional steps to the load history, the specified step and increment should be the last step and last increment of the previous analysis. Alternatively, they can be omitted and by default Abaqus will read the last available data in the restart file. The *RESTART option is followed by any new step definitions.

```
*HEADING
Add new step data
*RESTART, READ, STEP=<last step>, INC=<last increment>
*STEP
... new step definition...
*END STEP
```

Changing an analysis

Sometimes, having viewed the results of the previous analysis, you may want to restart the analysis from an intermediate point and change the remaining load history data in some manner—for example, to add more output requests, to change the loading, or to adjust the analysis controls. Often this is necessary when a step has exceeded its maximum number of increments. If the analysis is restarted as described above, Abaqus thinks that the analysis is partway through a step, tries to complete the step, and promptly exceeds the maximum number of increments again.

In such situations the END STEP parameter should be included on the *RESTART option to indicate that the current step should be terminated at the step and increment specified on the *RESTART option and all previously defined history data should be ignored. The simulation may then continue with the new steps defined after the *RESTART option. For example, if a step allowed only a maximum of 20 increments, which was less than the number of increments necessary to complete the step, the following restart input file would allow Abaqus to restart the simulation and finish the applied load:

```
*HEADING
Continue an analysis that exceeded the maximum number of
increments
```

```
*RESTART, READ, STEP=<step>, INC=20, END STEP
*STEP, INC=100
... repeat step definition...
*END STEP
```

In this situation the entire step definition, including applied loads and boundary conditions, should be identical to that specified in the original run with the following exceptions:

- The number of increments should be increased.
- The total time of the new step should be the total time of the original step less the time completed in the step in the first run. For example, if the time of the step as originally specified was 100 seconds and the analysis ran out of increments at a step time of 20 seconds, the duration of the step in the restart analysis should be 80 seconds.
- Any amplitude definitions specified in terms of step time need to be respecified to reflect the new time scale of the step. Amplitude definitions specified in terms of total time do not need to be changed, provided the modifications given above are used.

The magnitudes of any loads or prescribed boundary conditions remain unchanged since they are always total values in general analysis steps.

11.5 Example: restarting the pipe vibration analysis

To demonstrate how to restart an analysis, take the pipe section example in "Example: vibration of a piping system," Section 11.3, and restart the simulation, adding two additional steps of load history. The first simulation predicted that the piping section would be vulnerable to resonance when extended axially; you must now determine how much additional axial load will increase the pipe's lowest vibrational frequency to an acceptable level.

Step 3 will be a general step that increases the axial load on the pipe to 8 MN, and Step 4 will calculate the eigenmodes and eigenfrequencies again.

Create a new input file, called **pipe-2.inp**, and add the option blocks discussed below. If you wish to create the entire model using Abaqus/CAE, refer to "Example: restarting the pipe vibration analysis," Section 11.5 of Getting Started with Abaqus: Interactive Edition.

11.5.1 Reviewing the input file—the model data

The only model data required are the *HEADING option and a *RESTART option to read the restart data from the end of the previous analysis. Abaqus reads all other model data, such as node and element definitions, directly from the restart file. Add the following option blocks to your new input file:

```
*HEADING
Increase tensile load on the piping system
```

```
and determine lowest frequency.
*RESTART, READ
```

Neither the INCREMENT nor the STEP parameter is included on the *RESTART, READ option since by default Abaqus will read the data for the last increment written to the restart file. Since you are continuing the simulation from the end of the previous analysis, no parameters are needed.

11.5.2 Reviewing the input file—the history data

The history data consist of two steps. Apply a tensile load (8 MN) to the pipe section in Step 3. The following option block must be placed in Step 3:

```
*CLOAD
RIGHT, 1, 8.0E6
```

Set the initial time increment in Step 3 to 1/10 the total step time, which should be 1.0. Step 4 is an exact copy of Step 2 from the previous analysis. All of the load history option blocks necessary to define this restart analysis are shown below.

```
*STEP, NLGEOM=YES
Apply 8 MN axial tensile load
*STATIC
0.1, 1.
*CLOAD
RIGHT, 1, 8.0E6
*RESTART, WRITE, FREQUENCY=10
*OUTPUT, FIELD, FREQUENCY=10, VARIABLE=PRESELECT
*OUTPUT, HISTORY
*ELEMENT OUTPUT, ELSET=ELEMENT25
S, SINV
*END STEP
*STEP, PERTURBATION
Extract modes and frequencies
*FREQUENCY
8,
*RESTART, WRITE
*OUTPUT, FIELD, VARIABLE=PRESELECT
*END STEP
```

The complete input file for this restart analysis is listed in “Vibration of a piping system,” Section A.12.

11.5.3 Running the analysis

When running a simulation that will need to read data from a restart file, you must specify the root name of the restart file, without the **.res** extension, with the **oldjob** parameter on the Abaqus command line. Thus, use the following command to run this restart analysis:

```
abaqus job=pipe-2 oldjob=pipe
```

11.5.4 Status file

Again, check the status file as the job is running. When the analysis completes, the contents of the status file will look like

```
SUMMARY OF JOB INFORMATION:
STEP   INC ATT SEVERE EQUIL TOTAL  TOTAL      STEP       INC OF       DOF    IF
               DISCON ITERS ITERS  TIME/     TIME/LPF    TIME/LPF    MONITOR RIKS
               ITERS               FREQ
  3     1   1     0     1     1  1.10       0.100       0.1000
  3     2   1     0     1     1  1.20       0.200       0.1000
  3     3   1     0     1     1  1.35       0.350       0.1500
  3     4   1     0     1     1  1.58       0.575       0.2250
  3     5   1     0     1     1  1.91       0.913       0.3375
  3     6   1     0     1     1  2.00       1.00        0.08750
  4     1   1     0     6     0  2.00       1.00e-36    1.000e-36
```

This analysis starts at Step 3 since Steps 1 and 2 were completed in the previous analysis. There are now two output database (**.odb**) files associated with this simulation. Data for Steps 1 and 2 are in the file **pipe.odb**; data for Steps 3 and 4 are in the file **pipe-2.odb**. When plotting results in Abaqus/Viewer, you need to remember which results are stored in each file, and you need to ensure that Abaqus/Viewer is using the correct output database file.

11.5.5 Postprocessing the restart analysis results

Start Abaqus/Viewer and specify that the output database file from the restart analysis should be used by giving the following command:

```
abaqus viewer odb=pipe-2
```

Plotting the eigenmodes of the pipe

Plot the same six eigenmode shapes of the pipe section for this simulation as were plotted in the previous analysis. The eigenmode shapes can be plotted using the procedures described for the original analysis. These eigenmodes and their natural frequencies are shown in Figure 11–12; again, the corresponding mode shapes lie in planes orthogonal to each other.

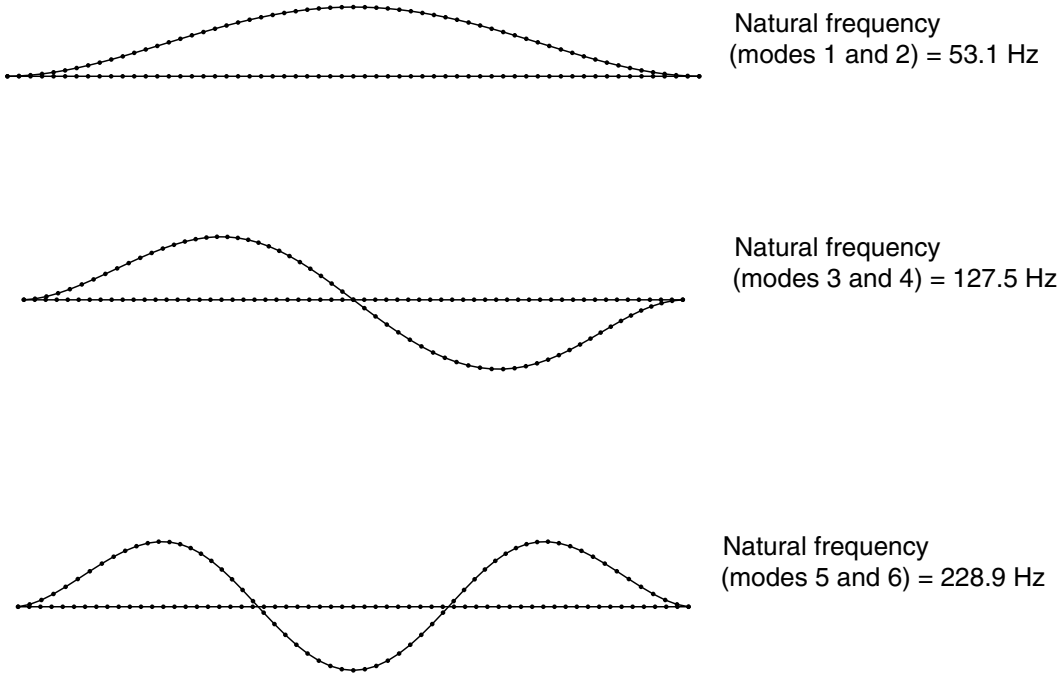


Figure 11–12 Shapes and frequencies of eigenmodes 1 through 6 with 8 MN tensile load.

Under 8 MN of axial load, the lowest mode is now at 53.1 Hz, which is greater than the required minimum of 50 Hz. If you want to find the exact load at which the lowest mode is just above 50 Hz, you can repeat this restart analysis and change the value of the applied load.

Plotting *X–Y* graphs from field data for selected steps

Use the field data stored in the output database files, **`pipe.odb`** and **`pipe-2.odb`**, to plot the history of the axial stress in the pipe for the whole simulation.

To generate a history plot of the axial stress in the pipe for the restart analysis:

1. In the Results Tree, double-click **XYData**.
   The **Create XY Data** dialog box appears.
2. Select **ODB field output** from this dialog box, and click **Continue** to proceed.
   The **XY Data from ODB Field Output** dialog box appears.
3. In the **Variables** tabbed page of this dialog box, accept the default selection of **`Integration Point`** for the variable position and select **S11** from the list of available stress components.

4. At the bottom of the dialog box, toggle **Select** for the section point and click **Settings** to choose a section point.
5. In the **Field Report Section Point Settings** dialog box that appears, select the category **beam** and choose any of the available section points for the pipe cross-section. Click **OK** to exit this dialog box.
6. In the **Elements/Nodes** tabbed page of the **XY Data from ODB Field Output** dialog box, select **Element labels** as the selection **Method**. There are 30 elements in the model, and they are numbered consecutively from 1 to 30. Enter any element number (for example, **25**) in the **Element labels** text field that appears on the right side of the dialog box.
7. Click **Active Steps/Frames**, and select **Step-3** as the only step from which to extract data.
8. At the bottom of the **XY Data from ODB Field Output** dialog box, click **Plot** to see the history of axial stress in the element.

The plot traces the axial stress history at each integration point of the element in the restart analysis. Since the restart is a continuation of an earlier job, it is often useful to view the results from the entire (original and restarted) analysis.

To generate a history plot of the axial stress in the pipe for the entire analysis:

1. Save the current plot by clicking **Save** at the bottom of the **XY Data from ODB Field Output** dialog box. Two curves are saved (one for each integration point), and default names are given to the curves.
2. Rename either curve **RESTART**, and delete the other curve.
3. From the main menu bar, select **File→Open**; or use the tool in the **File** toolbar to open the file **pipe.odb**.
4. Following the procedure outlined above, save the plot of the axial stress history for the same element and integration/section point used above. Name this plot **ORIGINAL**.
5. In the Results Tree, expand the **XYData** container.
   The **ORIGINAL** and **RESTART** curves are listed underneath.
6. Select both plots with [Ctrl]+Click. Click mouse button 3, and select **Plot** from the menu that appears to create a plot of axial stress history in the pipe for the entire simulation.
7. To change the style of the line, open the **Curve Options** dialog box.
8. For the **RESTART** curve, select a dotted line style.
9. Click **Dismiss** to close the dialog box.
10. To change the axis titles, open the **Axis Options** dialog box.
    In this dialog box, switch to the **Title** tabbed page.
11. Change the *X*-axis title to **TOTAL TIME**, and change the *Y*-axis title to **STRESS S11**.

12. Click **Dismiss** to close the dialog box.

The plot created by these commands is shown in Figure 11–13. The axial stress history of the same element during Step 3 can be plotted by itself by selecting only the **RESTART** curve (see Figure 11–14).

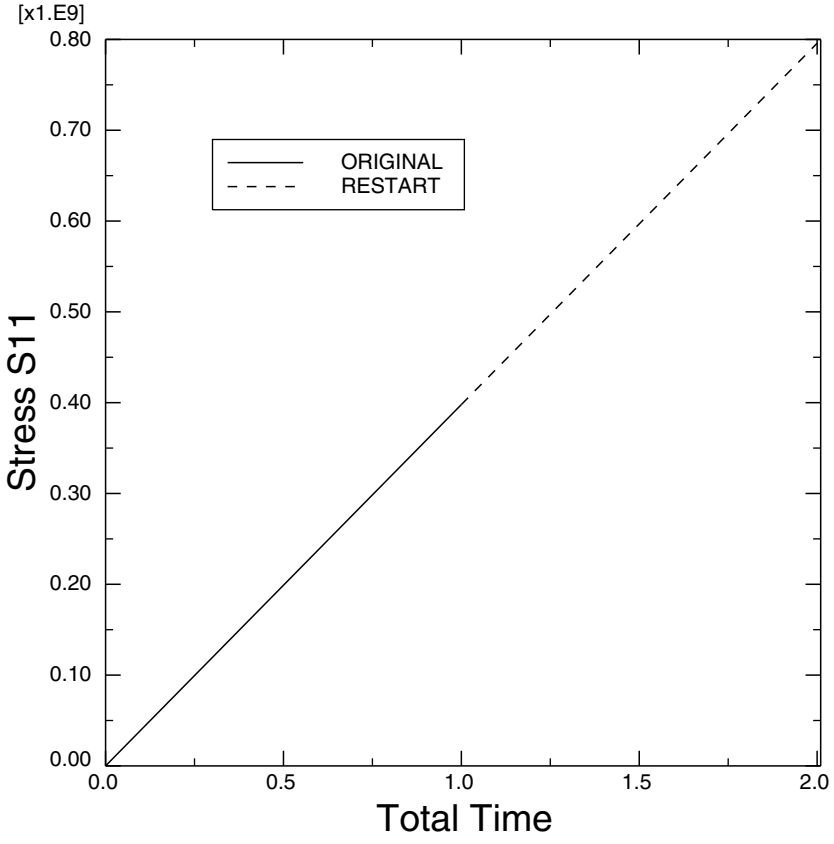


Figure 11–13 History of axial stress in the pipe.

11.6 Related Abaqus examples

- "Deep drawing of a cylindrical cup," Section 1.3.4 of the Abaqus Example Problems Manual

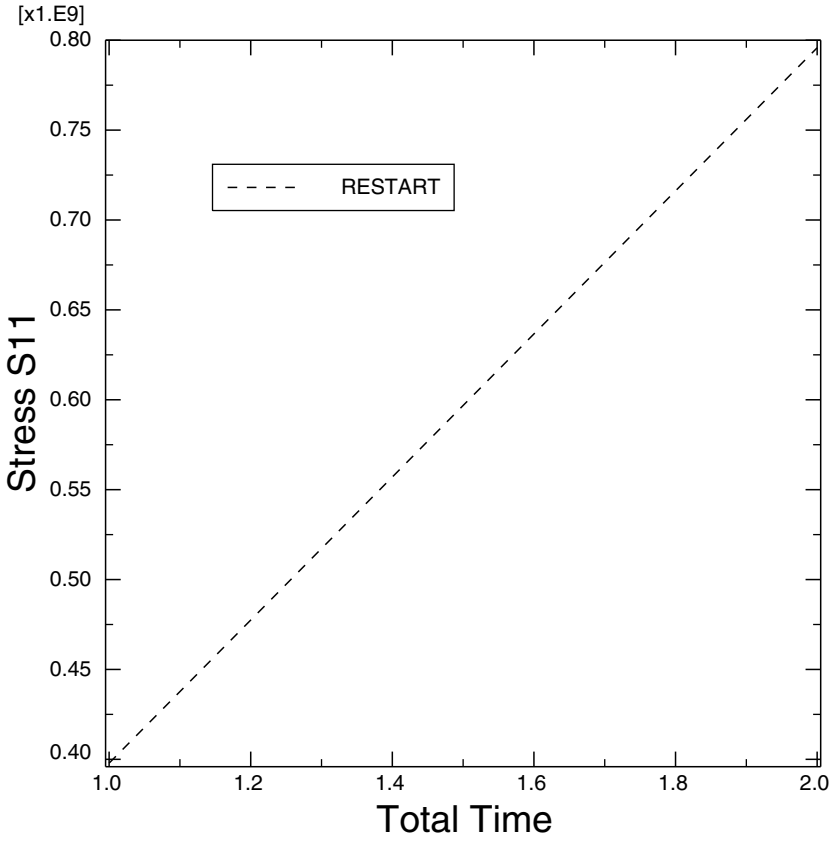


Figure 11–14 History of axial stress in the pipe during Step 3.

- "Linear analysis of the Indian Point reactor feedwater line," Section 2.2.2 of the Abaqus Example Problems Manual
- "Vibration of a cable under tension," Section 1.4.3 of the Abaqus Benchmarks Manual
- "Random response to jet noise excitation," Section 1.4.10 of the Abaqus Benchmarks Manual

11.7 Summary

- An Abaqus simulation can include any number of steps.
- Implicit and explicit steps are not allowed in the same analysis job.

- An analysis step is a period of "time" during which the response of the model to a specified set of loads and boundary conditions is calculated. The character of this response is determined by the particular analysis procedure used during the step.
- The response of a structure in a general analysis step may be either linear or nonlinear.
- The starting condition for each general step is the ending condition of the previous general step. Thus, the model's response evolves during a sequence of general steps in a simulation.
- Linear perturbation steps (available only in Abaqus/Standard) calculate the linear response of the structure to a perturbation load. The response is reported relative to the base state defined by the condition of the model at the end of the last general step.
- In general steps the OP parameter on any loading options—such as *BOUNDARY, *CLOAD, and *DLOAD—controls how the values specified with these options interact with those values defined in previous steps.
- Analyses can be restarted as long as a restart file is saved. Restart files can be used to continue an interrupted analysis or to add additional load history to the simulation.

12. Contact

Many engineering problems involve contact between two or more components. In these problems a force normal to the contacting surfaces acts on the two bodies when they touch each other. If there is friction between the surfaces, shear forces may be created that resist the tangential motion (sliding) of the bodies. The general aim of contact simulations is to identify the areas on the surfaces that are in contact and to calculate the contact pressures generated.

In a finite element analysis contact conditions are a special class of discontinuous constraint, allowing forces to be transmitted from one part of the model to another. The constraint is discontinuous because it is applied only when the two surfaces are in contact. When the two surfaces separate, no constraint is applied. The analysis has to be able to detect when two surfaces are in contact and apply the contact constraints accordingly. Similarly, the analysis must be able to detect when two surfaces separate and remove the contact constraints.

12.1 Overview of contact capabilities in Abaqus

Contact simulations in Abaqus/Standard can either be surface based or contact element based. Contact simulations in Abaqus/Explicit are surface based only. In this manual, surface-based contact is discussed.

Surface-based contact can utilize either the general ("automatic") contact algorithm or the contact pair algorithm. The general contact algorithm allows for a highly automated contact definition, where contact is based on an automatically generated all-inclusive surface definition. Conversely, the contact pair algorithm requires you to explicitly pair surfaces that may potentially come into contact. Both algorithms require specification of contact properties between surfaces (for example, friction).

The discussion of contact in this manual addresses the contact pair approach and general contact in Abaqus/Standard and general contact in Abaqus/Explicit.

12.2 Interaction between surfaces

The interaction between contacting surfaces consists of two components: one normal to the surfaces and one tangential to the surfaces. The tangential component consists of the relative motion (sliding) of the surfaces and, possibly, frictional shear stresses. Each contact interaction can refer to a contact property that specifies a model for the interaction between the contacting surfaces. There are several contact interaction models available in Abaqus; the default model is frictionless contact with no bonding.

12.2.1 Behavior normal to the surfaces

The distance separating two surfaces is called the clearance. The contact constraint is applied in Abaqus when the clearance between two surfaces becomes zero. There is no limit in the contact formulation on the magnitude of contact pressure that can be transmitted between the surfaces. The surfaces separate when the contact pressure between them becomes zero or negative, and the constraint is removed. This behavior, referred to as "hard" contact, is the default contact behavior in Abaqus and is summarized in the contact pressure-clearance relationship shown in Figure 12–1.

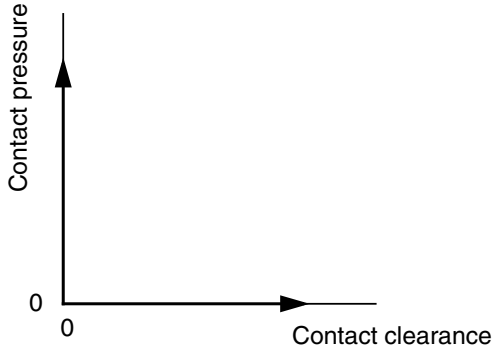


Figure 12–1 Contact pressure-clearance relationship for "hard" contact.

By default, "hard" contact is directly enforced when using contact pairs in Abaqus/Standard. The dramatic change in contact pressure that occurs when a contact condition changes from "open" (a positive clearance) to "closed" (clearance equal to zero) sometimes makes it difficult to complete contact simulations in Abaqus/Standard; the same is not true for Abaqus/Explicit since iteration is not required for explicit methods. Alternative enforcement methods (e.g., penalty) are available for contact pairs, as discussed in "Contact constraint enforcement methods in Abaqus/Standard," Section 37.1.2 of the Abaqus Analysis User's Manual. Penalty enforcement of the contact constraints is the only option available for general contact. Other sources of information include "Common difficulties associated with contact modeling in Abaqus/Standard," Section 38.1.2 of the Abaqus Analysis User's Manual; "Common difficulties associated with contact modeling using contact pairs in Abaqus/Explicit," Section 38.2.2 of the Abaqus Analysis User's Manual; the "Modeling Contact with Abaqus/Standard" lecture notes; and the "Advanced Topics: Abaqus/Explicit" lecture notes.

12.2.2 Sliding of the surfaces

In addition to determining whether contact has occurred at a particular point, an Abaqus analysis also must calculate the relative sliding of the two surfaces. This can be a very complex calculation; therefore,

Abaqus makes a distinction between analyses where the magnitude of sliding is small and those where the magnitude of sliding may be finite. It is much less expensive computationally to model problems where the sliding between the surfaces is small. What constitutes "small sliding" is often difficult to define, but a general guideline to follow is that problems can use the "small-sliding" approximation if a point contacting a surface does not slide more than a small fraction of a typical element dimension.

Small sliding is not available for general contact.

12.2.3 Friction models

When surfaces are in contact, they usually transmit shear as well as normal forces across their interface. Thus, the analysis may need to take frictional forces, which resist the relative sliding of the surfaces, into account. *Coulomb friction* is a common friction model used to describe the interaction of contacting surfaces. The model characterizes the frictional behavior between the surfaces using a coefficient of friction, μ.

The default friction coefficient is zero. The tangential motion is zero until the surface traction reaches a critical shear stress value, which depends on the normal contact pressure, according to the following equation:

$$\tau_{\text{crit}} = \mu p,$$

where μ is the coefficient of friction and p is the contact pressure between the two surfaces. This equation gives the limiting frictional shear stress for the contacting surfaces. The contacting surfaces will not slip (slide relative to each other) until the shear stress across their interface equals the limiting frictional shear stress, μp. For most surfaces μ is normally less than unity. Coulomb friction can be defined with μ or τ_{crit}. The solid line in Figure 12–2 summarizes the behavior of the Coulomb friction model: there is zero relative motion (slip) of the surfaces when they are sticking (the shear stresses are below μp). Optionally, a friction stress limit can be specified if both contacting surfaces are element-based surfaces.

In Abaqus/Standard the discontinuity between the two states—sticking or slipping—can result in convergence problems during the simulation. You should include friction in your Abaqus/Standard simulations only when it has a significant influence on the response of the model. If your contact simulation with friction encounters convergence problems, one of the first modifications you should try in diagnosing the difficulty is to rerun the analysis without friction. In general, friction presents no additional computational difficulties for Abaqus/Explicit.

Simulating ideal friction behavior can be very difficult; therefore, by default in most cases, Abaqus uses a penalty friction formulation with an allowable "elastic slip," shown by the dotted line in Figure 12–2. The "elastic slip" is the small amount of relative motion between the surfaces that occurs when the surfaces should be sticking. Abaqus automatically chooses the penalty stiffness (the slope of the dotted line) so that this allowable "elastic slip" is a very small fraction of the characteristic element length. The penalty friction formulation works well for most problems, including most metal forming applications.

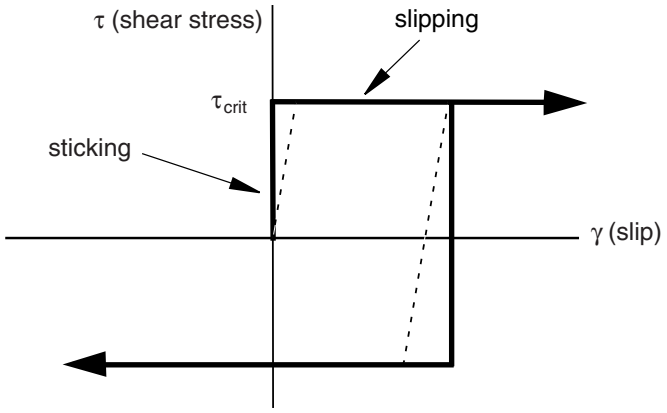


Figure 12–2 Frictional behavior.

In those problems where the ideal stick-slip frictional behavior must be included, the "Lagrange" friction formulation can be used in Abaqus/Standard and the kinematic friction formulation can be used in Abaqus/Explicit. The "Lagrange" friction formulation is more expensive in terms of the computer resources used because Abaqus/Standard uses additional variables for each surface node with frictional contact. In addition, the solution converges more slowly so that additional iterations are usually required. This friction formulation is not discussed in this guide.

Often the friction coefficient at the initiation of slipping from a sticking condition is different from the friction coefficient during established sliding. The former is typically referred to as the static friction coefficient, and the latter is referred to as the kinetic friction coefficient. In Abaqus an exponential decay law is available to model the transition between static and kinetic friction (see Figure 12–3). This friction formulation is not discussed in this guide.

In Abaqus/Standard the inclusion of friction in a model adds unsymmetric terms to the system of equations being solved. If μ is less than about 0.2, the magnitude and influence of these terms are quite small and the regular, symmetric solver works well (unless the contact surface has high curvature). For higher coefficients of friction, the unsymmetric solver is invoked automatically because it will improve the convergence rate. The unsymmetric solver requires twice as much computer memory and scratch disk space as the symmetric solver. The unsymmetric solver can also be selected by including the UNSYMM=YES parameter on the ∗STEP option. Large values of μ generally do not cause any difficulties in Abaqus/Explicit.

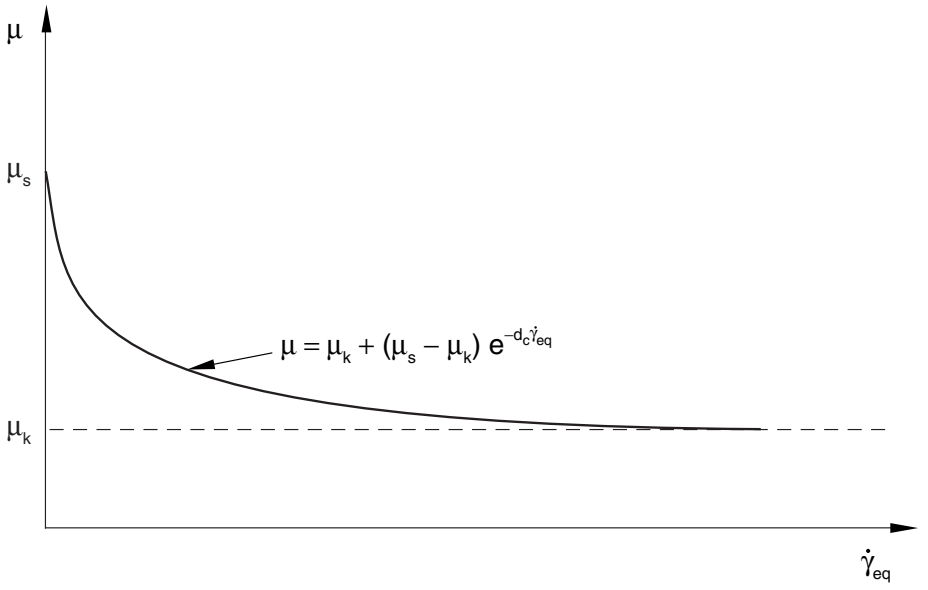


Figure 12–3 Exponential decay friction model.

12.2.4 Other contact interaction options

The other contact interaction models available in Abaqus depend on the analysis product and the algorithm used and may include adhesive contact behavior, softened contact behavior, fasteners (for example, spot welds), and viscous contact damping. These options are not discussed in this guide. Details about them can be found in the Abaqus Analysis User's Manual.

12.2.5 Surface-based constraints

Tie constraints are used to tie together two surfaces for the duration of a simulation. Each node on the slave surface is constrained to have the same motion as the point on the master surface to which it is closest. For a structural analysis this means the translational (and, optionally, the rotational) degrees of freedom are constrained.

Abaqus uses the undeformed configuration of the model to determine which slave nodes are tied to the master surface. By default, all slave nodes that lie within a given distance of the master surface are tied. The default distance is based on the typical element size of the master surface. This default can be overridden in one of two ways: by specifying the distance within which slave nodes must lie from the master surface to be constrained or by specifying the name of a set containing the nodes that will be constrained.

Slave nodes can also be adjusted so that they lie exactly on the master surface. If slave nodes have to be adjusted by distances that are a large fraction of the length of the side of the element (to which the slave node is attached), the element can become severely distorted; avoid large adjustments if possible.

Tie constraints are particularly useful for rapid mesh refinement between dissimilar meshes.

12.3 Defining contact in Abaqus/Standard

The first step in defining contact pairs in Abaqus/Standard is to create the surfaces using the *SURFACE option. The next step is to specify the surfaces that interact with one another using the *CONTACT PAIR option. Each contact interaction refers to a surface interaction definition, which is created with the *SURFACE INTERACTION option. A contact pressure-clearance relationship and friction properties can be assigned to a surface interaction definition.

The definition of surfaces is optional for general contact because an all-inclusive element-based surface is automatically created when the *CONTACT option is used. Specific surface pairings may be used, however, to include regions not included in the default surface (*CONTACT INCLUSIONS), to preclude interaction between different regions of a model (*CONTACT EXCLUSIONS), or to override global contact property assignments. For example, if you want to apply a certain friction coefficient to all but a few surfaces in your model, you can assign a global friction coefficient and and override this property for a given pair of user-defined surfaces using the *CONTACT PROPERTY ASSIGNMENT option.

12.3.1 Defining surfaces

Surfaces are created with the *SURFACE option by identifying all of the element faces that form the surface. This is done in much the same way as defining distributed pressure loads.

Surfaces on continuum elements

A two-dimensional, first-order continuum element, such as CPE4, has four faces consisting of the segments defined by nodes 1–2, 2–3, 3–4, and 4–1, respectively, as shown in Figure 12–4. The face identifiers consist of the letter “S” followed by the face number. For example, use the following option block to include face 2 of the element shown in Figure 12–4 in a surface called **FLANGE1**:

```
*SURFACE, NAME=FLANGE1
5, S2
```

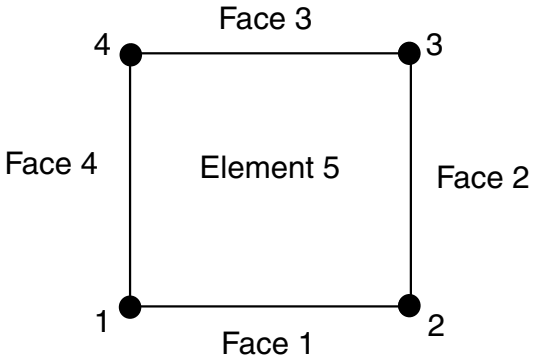


Figure 12–4 Face numbers on a two-dimensional, first-order (4-node) element.

As is the case for many options in Abaqus, both element numbers and element sets can be used; the use of element sets can make the definition of large surfaces much easier. It is valid to specify both element sets and individual elements in the same *SURFACE option block, as shown in Figure 12–5 and the example below. The surface **TOPSURF** consists of the element faces shown in Figure 12–5 and is created as follows:

```
*ELSET, ELSET=TOP, GENERATE
5, 8
*SURFACE, NAME=TOPSURF
TOP, S3
5, S4
8, S2
```

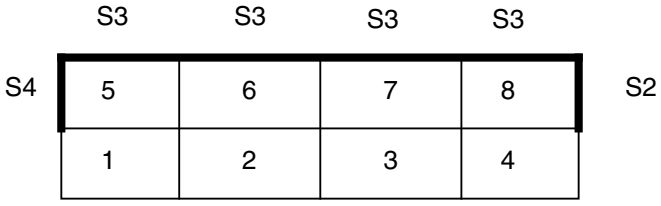


Figure 12–5 Face numbers and elements that form the surface **TOPSURF**.

Abaqus can determine the free faces of two- and three-dimensional continuum elements automatically and use them to create a surface. To use this capability, simply include all the elements whose free faces make up the surface on the data lines of the *SURFACE option. Either

element sets or individual element numbers can be used. If elements in the interior of the body are included, Abaqus will ignore them. For example, the surface shown in Figure 12–5 could also be defined using

```
*SURFACE, NAME=TOPSURF
TOP,
```

Surfaces on shell, membrane, and rigid elements

For shell, membrane, and rigid elements you must specify which side of the element forms the contact surface. The side in the direction of the positive element normal is called SPOS, while the side in the direction of the negative element normal is called SNEG, as shown in Figure 12–6. As discussed in Chapter 5, "Using Shell Elements," the connectivity of an element defines the positive element normal. The positive element normals can be viewed in Abaqus/Viewer.

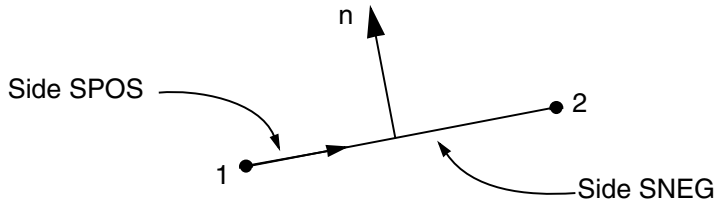


Figure 12–6 Surfaces on a two-dimensional shell or rigid element.

The following option block defines surface **SURF1** as the surface composed of all the SPOS faces of the elements in element set **SHELLS**:

```
*SURFACE, NAME=SURF1
SHELLS, SPOS
```

Restrictions on the types of surfaces that can be created in Abaqus are discussed in "Surface definition," Section 2.3 of the Abaqus Analysis User's Manual; please read them before beginning a contact simulation.

Rigid surfaces

Rigid surfaces are the surfaces of rigid bodies. They can be defined as an analytical shape, or they can be based on the underlying surfaces of elements associated with the rigid body.

Analytical rigid surfaces are created by defining a series of connected lines, arcs, and parabolas. The parameter ANALYTICAL SURFACE on the *RIGID BODY option binds an analytical rigid surface (defined with the TYPE parameter on the *SURFACE option) with a rigid body. The *RIGID BODY option must be defined in the model definition. The TYPE parameter on the *SURFACE option defines the dimensionality of the surface, and it has three possible values:

- Use TYPE=SEGMENTS to define a two-dimensional analytical rigid surface.
- Use TYPE=CYLINDRICAL to define a three-dimensional analytical rigid surface that is extruded infinitely in the out-of-plane direction.
- Use TYPE=REVOLUTION to define a three-dimensional analytical rigid surface of revolution.

The following is an example input for the two-dimensional analytical rigid surface named **SRIGID** shown in Figure 12–7:

```
*SURFACE, TYPE=SEGMENTS, NAME=SRIGID
START, 5.0, 0.0
LINE, 10.0, 0.0
CIRCL, 15.0, 5.0, 10.0, 5.0
```

where the rigid body is defined by

```
*RIGID BODY, ANALYTICAL SURFACE=SRIGID, REF NODE=10000
```

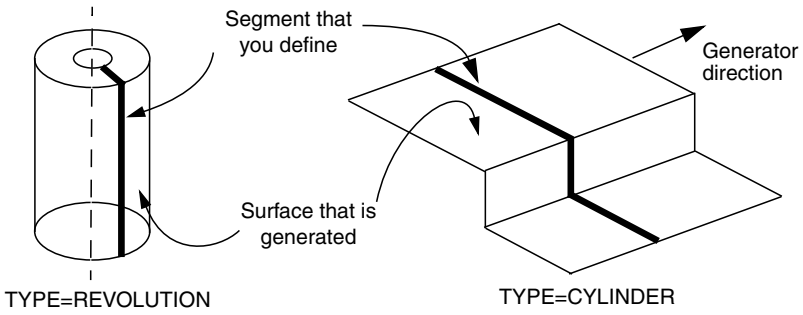


Figure 12–7 Analytical rigid surfaces.

Discretized rigid surfaces are based on the underlying elements that make up a rigid body; thus, they can be more geometrically complex than analytical rigid surfaces. Discretized rigid surfaces are defined using the *SURFACE option in exactly the same manner as surfaces on deformable bodies.

12.3.2 Contact interactions

With the contact pair approach, you define possible contact between two surfaces in an Abaqus/Standard simulation by giving the surface names on the *CONTACT PAIR option. When you define a contact pair, you must decide whether the magnitude of the relative sliding will be small or finite. The default is the more general finite-sliding formulation. The small-sliding formulation is appropriate if the relative

motion of the two surfaces is less than a small proportion of the characteristic length of an element face. The small-sliding formulation is selected by including the SMALL SLIDING parameter on the ∗CONTACT PAIR option. Using the small-sliding formulation will result in a more efficient analysis.

Each contact pair must refer to a surface interaction definition, in much the same way that each element must refer to an element property definition. Use the INTERACTION parameter on the ∗CONTACT PAIR option to refer to a ∗SURFACE INTERACTION option where the different surface interaction models, such as ∗FRICTION, can be defined.

The following example:

```
*CONTACT PAIR, INTERACTION=FRIC, SMALL SLIDING
FLANGE1, FLANGE2
*SURFACE INTERACTION, NAME=FRIC
*FRICTION
0.1,
```

specifies that surfaces **FLANGE1** and **FLANGE2** might interact with each other and that the amount of relative sliding that occurs will be considered to be small. The interaction between the surfaces includes friction, with a friction coefficient of 0.1.

With the general contact approach, you do not need to explicitly define and pair individual surfaces. By using the ∗CONTACT option and its related sub-options, Abaqus/Standard automatically defines an "all-inclusive" element-based surface and enforces contact between the members of that surface. The contact definition as a result is considerably simplified. The following example:

```
*CONTACT
*CONTACT INCLUSIONS, ALL EXTERIOR
*CONTACT PROPERTY ASSIGNMENT
 , , FRIC,
*SURFACE INTERACTION, NAME=FRIC
*FRICTION
0.1
```

specifies that all exterior faces in a given model might interact with each other. The interaction between all surfaces includes friction, with a friction coefficient of 0.1.

12.3.3 Slave and master surfaces

By default, contact pairs in Abaqus/Standard use a pure master-slave contact algorithm: nodes on one surface (the slave) cannot penetrate the segments that make up the other surface (the master), as shown in Figure 12–8. The algorithm places no restrictions on the master surface; it can penetrate the slave surface between slave nodes, as shown in Figure 12–8. The order of the two surfaces given on the ∗CONTACT PAIR option determines which surface is the master surface and which is the slave surface; the first surface is the slave surface, and the second is the master surface.

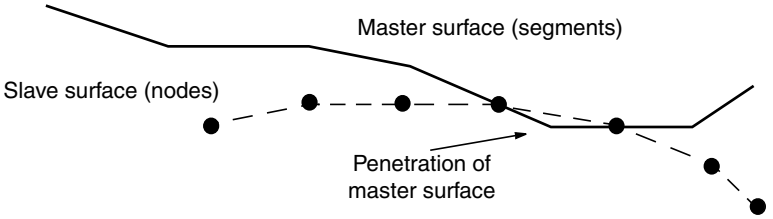


Figure 12–8 The master surface can penetrate the slave surface.

Due to the strict master-slave formulation, you must be careful to select the slave and master surfaces correctly in order to achieve the best possible contact simulation. Some simple rules to follow are:

- the slave surface should be the more finely meshed surface; and
- if the mesh densities are similar, the slave surface should be the surface with the softer underlying material.

The general contact algorithm in Abaqus/Standard enforces contact in an average sense between interacting surfaces; Abaqus/Standard automatically assigns master and slave roles.

12.3.4 Contact discretization

Abaqus/Standard offers two contact discretization methods: a traditional node-to-surface method and a surface-to-surface method. The node-to-surface discretization method defines contact conditions between each slave node and the master surface. The surface-to-surface discretization method considers the shape of both the master and slave surfaces when defining the contact constraints. The contact pair algorithm can use either discretization method; general contact uses only the surface-to-surface approach.

12.3.5 Small and finite sliding

When using the small-sliding formulation, Abaqus/Standard establishes the relationship between the slave nodes and the master surface at the beginning of the simulation. Abaqus/Standard determines which segment on the master surface will interact with each node on the slave surface. It maintains these relationships throughout the analysis, never changing which master surface segments interact with which slave nodes. If geometric nonlinearity is included in the model by setting the NLGEOM parameter equal to YES on the *STEP option, the small-sliding algorithm accounts for any rotation and deformation of the master surface and updates the load path through which the contact forces are transmitted. If geometric nonlinearity is not included in the model, any rotation or deformation of the master surface is ignored and the load path remains fixed.

The finite-sliding contact formulation requires that Abaqus/Standard continually track which part of the master surface is in contact with each slave node. This is a very complex calculation, especially if both the contacting bodies are deformable. The structures in such simulations can be either two- or three-dimensional. Abaqus/Standard can also model the finite-sliding self-contact of a deformable body. Such a situation occurs when a structure folds over onto itself.

The finite-sliding formulation for contact between a deformable body and a rigid surface is not as complex as the finite-sliding formulation for two deformable bodies. Finite-sliding simulations where the master surface is rigid can be performed for both two- and three-dimensional models.

The contact pair algorithm can consider either small or finite sliding effects; general contact only considers finite sliding effects.

12.3.6 Element selection

Selection of elements for contact depends heavily on the contact enforcement used. For example, for traditional contact formulations (i.e., the node-to-surface discretization) it is generally better to use first-order elements for those parts of a model that will form a slave surface. Second-order elements can sometimes cause problems in this case because of the way these elements calculate consistent nodal loads for a constant pressure. The consistent nodal loads for a constant pressure, P, on a second-order, two-dimensional element with area A are shown in Figure 12–9.

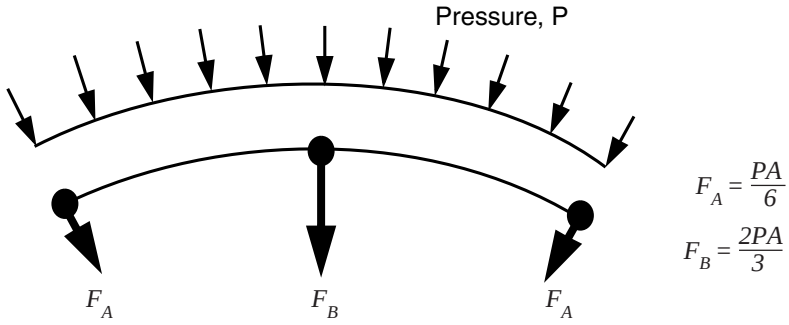


Figure 12–9 Equivalent nodal loads for a constant pressure on a two-dimensional, second-order element.

The node-to-surface contact formulation bases important decisions on the forces acting on the slave nodes. It is difficult for the algorithm to tell if the force distribution shown in Figure 12–9 represents a constant contact pressure or an actual variation across the element. The equivalent nodal forces for a three-dimensional, second-order brick element are even more confusing because they do not even have the same sign for a constant pressure, making it very difficult for the algorithm to work correctly, especially for nonuniform contact. Therefore, to avoid such problems, Abaqus/Standard automatically adds a midface node to any face of a second-order, three-dimensional brick or wedge element that defines

a slave surface when it is used with the node-to-surface formulation. The equivalent nodal forces for a second-order element face with a midface node have the same sign for a constant pressure, although they still differ considerably in magnitude.

The equivalent nodal forces for applied pressures on first-order elements always have a consistent sign and magnitude; therefore, there is no ambiguity about the contact state that a given distribution of nodal forces represents.

If you are using the node-to-surface formulation and your geometry is complicated and requires the use of an automatic mesh generator, the modified second-order tetrahedral elements (C3D10M) in Abaqus/Standard should be used. These elements are designed to be used in complex contact simulations; regular second-order tetrahedral elements (C3D10) have zero contact force at their corner nodes, leading to poor predictions of the contact pressures. The modified second-order tetrahedral elements can calculate the contact pressures accurately.

Regular second-order elements can generally be used without difficulty with the surface-to-surface formulation.

12.3.7 Contact algorithm

Understanding the algorithm Abaqus/Standard uses to solve contact problems will help you understand the diagnostic output in the message file and carry out contact simulations successfully.

The contact algorithm in Abaqus/Standard, which is shown in Figure 12–10, is built around the Newton-Raphson technique discussed in Chapter 8, "Nonlinearity." Abaqus/Standard examines the state of all contact interactions at the start of each increment to establish whether slave nodes are open or closed. If a node is closed, Abaqus/Standard determines whether it is sliding or sticking. Abaqus/Standard applies a constraint for each closed node and removes constraints from any node where the contact state changes from closed to open. Abaqus/Standard then carries out an iteration and updates the configuration of the model using the calculated corrections.

In the updated configuration Abaqus/Standard checks for changes in the contact conditions at the slave nodes. Any node where the clearance after the iteration becomes negative or zero has changed status from open to closed. Any node where the contact pressure becomes negative has changed status from closed to open. If any contact changes are detected in the current iteration, Abaqus/Standard labels it a *severe discontinuity iteration*.

Abaqus/Standard continues to iterate until the severe discontinuities are sufficiently small (or no severe discontinuities occur) and the equilibrium (flux) tolerances are satisfied. Alternatively, you can choose a different approach in which Abaqus/Standard will continue to iterate until no severe discontinuities occur before checking for equilibrium.

The summary for each completed increment in the message and status files shows how many iterations were severe discontinuity iterations and how many were equilibrium iterations (an equilibrium iteration is one in which no severe discontinuities occur). The total number of iterations for an increment is the sum of these two. For some increments, you may find that all iterations are labeled severe discontinuity iterations (this occurs when small contact changes are detected in each iteration and equilibrium is ultimately satisfied).

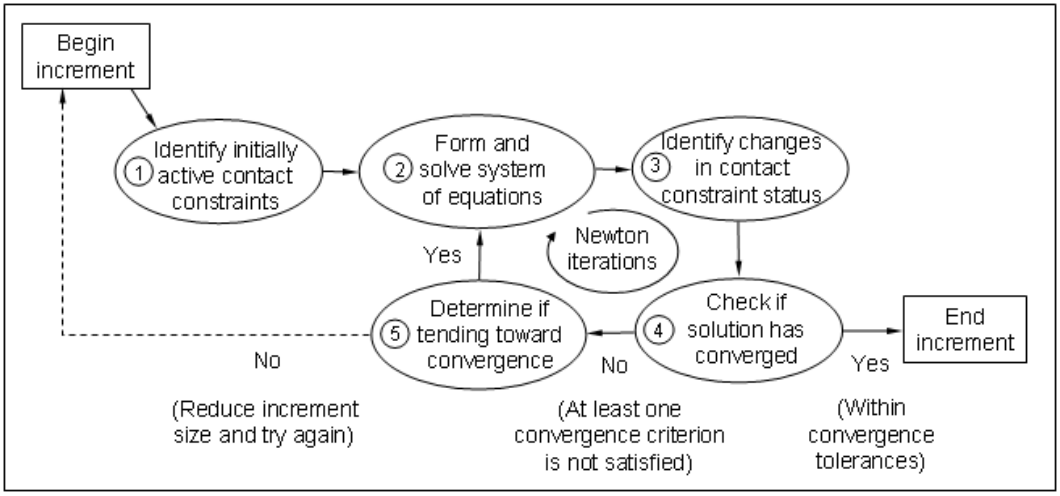


Figure 12–10 Contact algorithm in Abaqus/Standard.

Abaqus/Standard applies sophisticated criteria involving changes in penetration, changes in the residual force, and the number of severe discontinuities from one iteration to the next to determine whether iteration should be continued or terminated. Hence, it is in principle not necessary to limit the number of severe discontinuity iterations. This makes it possible to run contact problems that require large numbers of contact changes without having to change the control parameters. The default limit for the maximum number of severe discontinuity iterations is 50, which in practice should always be more than the actual number of iterations in an increment.

12.4 Modeling issues for rigid surfaces in Abaqus/Standard

There are a number of issues that you should consider when modeling contact problems in Abaqus/Standard that involve rigid surfaces. These issues are discussed in detail in "Common difficulties associated with contact modeling in Abaqus/Standard," Section 38.1.2 of the Abaqus Analysis User's Manual; but some of the more important issues are described here.

- The rigid surface is always the master surface in a contact interaction.
- The rigid surface should be large enough to ensure that slave nodes do not slide off and "fall behind" the surface. If this happens, the solution usually will fail to converge. Extending the rigid surface or including corners along the perimeter (see Figure 12–11) will prevent slave nodes from falling behind the master surface.

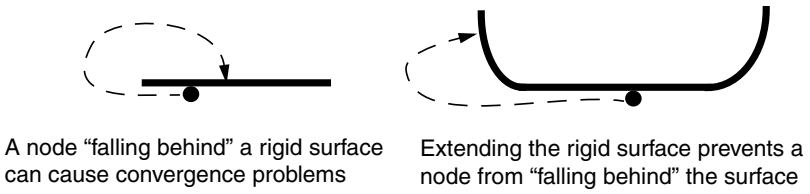

A node “falling behind” a rigid surface can cause convergence problems
Extending the rigid surface prevents a node from “falling behind” the surface

Figure 12–11 Extending rigid surfaces to prevent convergence problems.

- The deformable mesh must be refined enough to interact with any feature on the rigid surface. There is no point in having a 10 mm wide feature on the rigid surface if the deformable elements that will contact it are 20 mm across: the rigid feature will just penetrate into the deformable surface as shown in Figure 12–12.

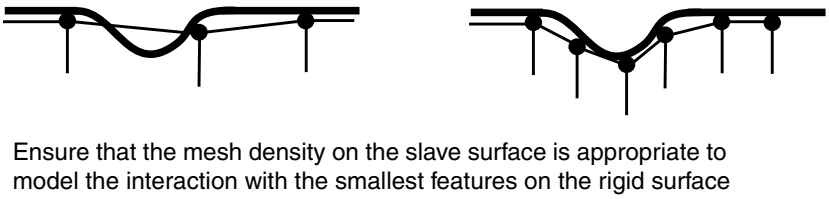

Ensure that the mesh density on the slave surface is appropriate to model the interaction with the smallest features on the rigid surface

Figure 12–12 Modeling small features on the rigid surface.

With a sufficiently refined mesh on the deformable surface, Abaqus/Standard will prevent the rigid surface from penetrating the slave surface.

- The contact algorithm in Abaqus/Standard requires the master surface of a contact pair to be smooth. Rigid surfaces are always the master surface and so should always be smoothed. Abaqus/Standard does not smooth discrete rigid surfaces. The level of refinement controls the smoothness of a discrete rigid surface. Analytical rigid surfaces can be smoothed using the FILLET RADIUS parameter on the *SURFACE option to define a fillet radius that is used to smooth any sharp corners in the rigid surface definition (see Figure 12–13.)
- The rigid surface normal must always point toward the deformable surface with which it will interact. If it does not, Abaqus/Standard will detect severe overclosures at all of the nodes on the deformable surface; the simulation will probably terminate due to convergence difficulties.

  The normals for an analytical rigid surface are defined as the directions obtained by the 90° counterclockwise rotation of the vectors from the beginning to the end of each line and circular segment forming the surface (see Figure 12–14).

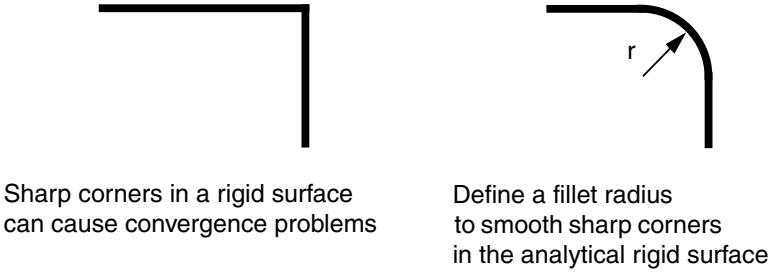

r
Sharp corners in a rigid surface
can cause convergence problems
Define a fillet radius
to smooth sharp corners
in the analytical rigid surface

Figure 12–13 Smoothing an analytical rigid surface.

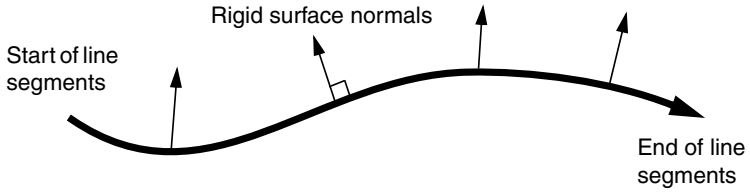


Figure 12–14 Normals for an analytical rigid surface.

The normals for a rigid surface created from rigid elements are defined by the faces specified on the *SURFACE option creating the surface.

12.5 Abaqus/Standard 2-D example: forming a channel

This simulation of the forming of a channel in a long metal sheet illustrates the use of rigid surfaces and some of the more complex techniques often required for a successful contact analysis in Abaqus/Standard.

The problem consists of a strip of deformable material, called the blank, and the tools—the punch, die, and blank holder—that contact the blank. The tools are modeled as (analytical) rigid surfaces because they are much stiffer than the blank. Figure 12–15 shows the basic arrangement of the components. The blank is 1 mm thick and is squeezed between the blank holder and the die. The blank holder force is 440 kN. This force, in conjunction with the friction between the blank and blank holder and the blank and die, controls how the blank material is drawn into the die during the forming process. You have been asked to determine the forces acting on the punch during the forming process. You also must assess how well the channel is formed with these particular settings for the blank holder force and the coefficient of friction between the tools and blank.

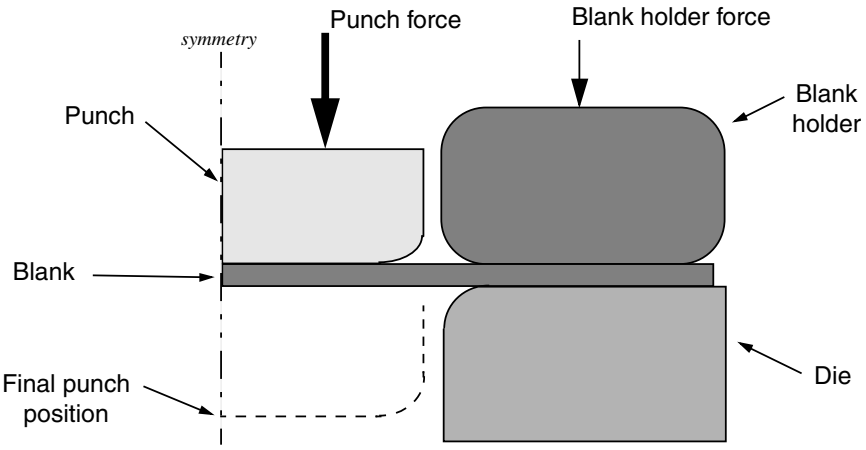


Figure 12–15 Forming analysis.

A two-dimensional, plane strain model will be used. The assumption that there is no strain in the out-of-plane direction of the model is valid if the structure is long in this direction. Only half of the channel needs to be modeled because the forming process is symmetric about a plane along the center of the channel.

The model will use contact pairs rather than general contact, since general contact is not available for analytical rigid surfaces in Abaqus/Standard.

The dimensions of the various components are shown in Figure 12–16.

12.5.1 Coordinate system

Two-dimensional, plane strain models are defined, by default, in the global 1–2 plane as shown in Figure 12–16. For the forming simulation, place the origin of this plane at the bottom left-hand corner of the blank (Figure 12–16). The 1-direction will be normal to the symmetry plane, which is located at $x = 0$.

12.5.2 Mesh design

The mesh for this simulation can be divided into the deformable blank and the rigid tools.

Blank

Once again, the element type should be selected before the mesh is designed. The mesh used for the blank should consist of four rows of 100 CPE4R elements (see Figure 12–17). Four rows of

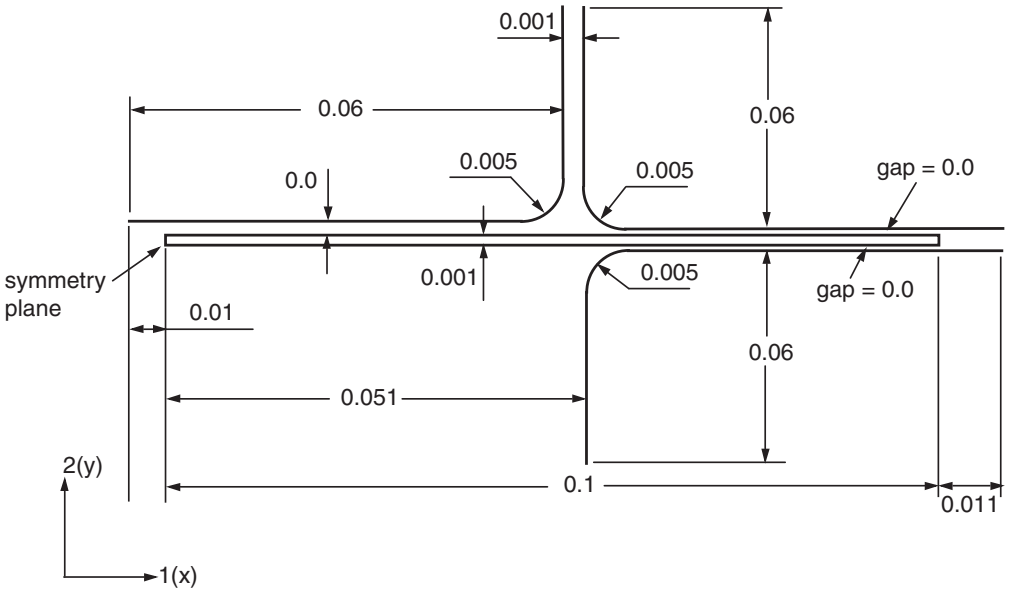


Figure 12–16 Dimensions, in m, of the components in the forming simulation.

elements are used so that better resolution of the deformation through the thickness of the blank will be obtained.

The node and element numbers for the mesh shown in Figure 12–18 are from the model of this problem given in "Forming a channel with Abaqus/Standard," Section A.13. These node and element numbers are used in the discussion that follows.

Tools

The tools are modeled with analytical rigid surfaces.

12.5.3 Preprocessing—creating the model

The steps that follow assume that you have access to the full input file for this example. This input file, **`channel.inp`**, is provided in "Forming a channel with Abaqus/Standard," Section A.13, in the online HTML version of this manual. Instructions on how to fetch and run the script are given in Appendix A, "Example Files." If you wish to create the entire model using Abaqus/CAE, please refer to "Abaqus/Standard 2-D example: forming a channel," Section 12.6 of Getting Started with Abaqus: Interactive Edition.

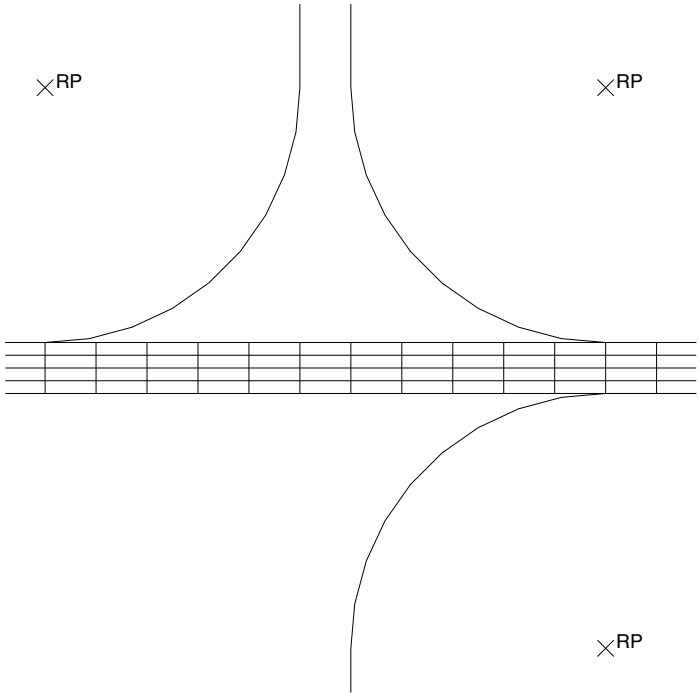


Figure 12–17 Mesh.

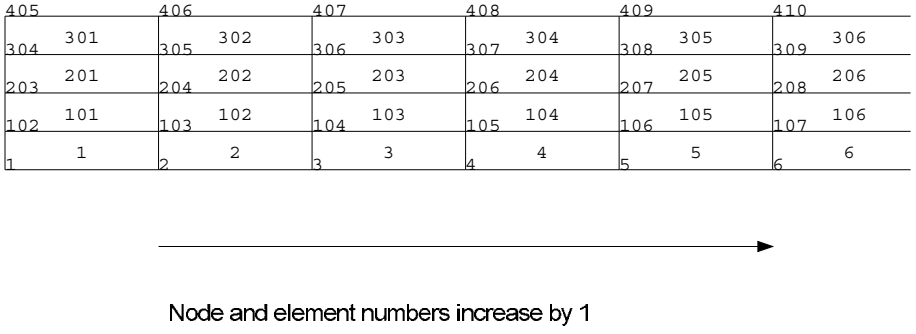


Figure 12–18 Node and element numbers for the blank.

12.5.4 Reviewing the input file—the model data

We first review the model definition, including the node and element definitions, and section and material properties.

Model description

The input file starts with a relevant description of the simulation and model in the *HEADING option.

```
*HEADING
Analysis of the forming of a channel
SI units (N, kg, m, s)
```

Nodal coordinates and element connectivity

Check that the preprocessor used the correct element type for the blank. Provide a meaningful element set name, such as **BLANK**, for the elements. The *ELEMENT option in this model follows:

```
*ELEMENT, TYPE=CPE4R, ELSET=BLANK
```

The model definition also specifies the creation of node sets so that parts of the model can be constrained and moved easily. These nodes are located on the centerline of the blank and have symmetric constraints into a node set called **CENTER**.

```
*NSET, NSET=CENTER
1,102,203,304,405
```

The node along the middle of the sheet at the left-hand side of the model, underneath the punch, is included in node set **MIDLEFT**.

```
*NSET, NSET=MIDLEFT
203,
```

Again, the node numbers in these option blocks are for the model in Figure 12–18; your node numbers may be different.

Two element sets, **BLANK_B** and **BLANK_T**, will be defined that contain the lower and upper rows of elements in the blank. These will be used to define the contact surfaces on the blank.

```
*ELSET, ELSET=BLANK_B, GENERATE
1,100,1
*ELSET, ELSET=BLANK_T, GENERATE
301,400,1
```

Section and material properties for the blank

The blank is made from a high-strength steel (elastic modulus of 210.0 GPa, $\nu = 0.3$). Its stress-strain behavior is shown in Figure 12–19. The material undergoes considerable work hardening as it deforms plastically. It is likely that plastic strains will be large in this analysis; therefore, hardening data are provided up to 50% plastic strain.

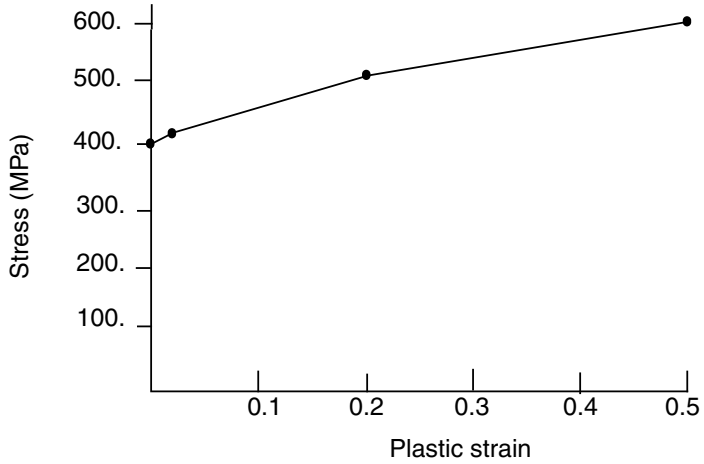


Figure 12–19 Yield stress vs. plastic strain.

The blank is going to undergo significant rotation as it deforms. Reporting the values of stress and strain in a coordinate system that rotates with the blank's motion will make it much easier to interpret the results. Therefore, an *ORIENTATION option should be used to create a coordinate system that is aligned initially with the global coordinate system but moves with the elements as they deform. The following input options are needed to define the blank's element and material properties:

```
*ORIENTATION, NAME=LOCAL
1.,0.,0., 0.,1.,0.
1, 0
*SOLID SECTION, MATERIAL=STEEL, ORIENTATION=LOCAL,
ELSET=BLANK, CONTROL=EC-1
*SECTION CONTROLS, NAME=EC-1, HOURGLASS=ENHANCED
*MATERIAL, NAME=STEEL
*ELASTIC
2.1E11,0.3
*PLASTIC
400.E6, 0.0E-2
```

```
420.E6, 2.0E-2
500.E6,20.0E-2
600.E6,50.0E-2
```

12.5.5 Contact definitions

The contact definitions for each part of the model are discussed here.

Rigid surfaces

The blank holder, the punch, and the die are modeled with analytical rigid surfaces. A rigid body reference node will be assigned to each of these surfaces when they are created. If you did not create these rigid body reference nodes during preprocessing, add the following option block to your model:

```
*NODE, NSET=REFPUNCH
7000, 0.000, 0.06
*NODE, NSET=REFHOLD
8000, 0.1,0.06
*NODE, NSET=REFDIE
9000,0.1,-0.06
*NSET, NSET=NOUT
REFDIE, REFHOLD, REFPUNCH
```

Each node is placed in a node set to make the input file easy to read. While someone unfamiliar with the specific mesh used in the simulation may not know why boundary conditions are applied to node 7000, they might be able to guess why boundary conditions were applied to the **REFPUNCH** node. All reference nodes are also assigned to a set named **NOUT** to facilitate the history output requests that will follow.

To ensure that an analytical rigid surface's normals point toward the deformable surfaces that the rigid surface will contact, the segments composing the rigid surface must be defined in a particular order. For example, to create the correct normals for the surface **PUNCH**, define the surface from the top right corner to the bottom left corner of the punch. The following input to the model creates the surface **PUNCH:**

```
*SURFACE, TYPE=SEGMENTS, NAME=PUNCH, FILLET RADIUS=0.001
START,0.050, 0.060
LINE, 0.050, 0.006
CIRCL,0.045, 0.001, 0.045,0.006
LINE,-0.010, 0.001
*RIGID BODY, ANALYTICAL SURFACE=PUNCH, REF NODE=7000
```

The parameter TYPE=SEGMENTS specifies that a two-dimensional rigid surface is being defined. The NAME parameter specifies the name of the surface, **PUNCH**. The data lines define the geometry of the surface. The first data line always has the word "START" followed by the 1- and 2-coordinates of the starting point for the surface. The subsequent lines define line, circular, and parabolic segments. For this surface the second data line defines a straight line from the start position (0.05, 0.060) to (0.050, 0.006). The third data line defines a circular arc from the end of the straight line (0.05, 0.006) to (0.045, 0.001) with the center of the circle located at (0.045, 0.006). The last data line defines a straight line from the end of the arc to (−0.010, 0.001).

This definition should produce a smooth rigid surface but, to be safe, the FILLET RADIUS parameter specifies that a 1 mm fillet radius should be used to smooth any discontinuities in the surface definition. It is always good practice to add the FILLET RADIUS parameter to the definition of any analytical rigid surface.

The *RIGID BODY option is used to bind the analytical surface to a rigid body with its rigid body reference node specified by the REF NODE parameter and the surface referred to by its name, using the ANALYTICAL SURFACE parameter.

The rigid surfaces for the blank holder and the die are defined in a similar way. The following option blocks define the surfaces on these tools:

```
*SURFACE, TYPE=SEGMENTS, NAME=HOLDER, FILLET RADIUS=0.001
START,0.110, 0.001
LINE, 0.056,0.001
CIRCL,0.051,0.006, 0.056,0.006
LINE, 0.051,0.060
*RIGID BODY, ANALYTICAL SURFACE=HOLDER, REF NODE=8000
**
*SURFACE, TYPE=SEGMENTS, NAME=DIE, FILLET RADIUS=0.001
START,0.051,-0.060
LINE, 0.051,-0.005
CIRCL,0.056,0.,0.056,-0.005
LINE, 0.11, 0.
*RIGID BODY, ANALYTICAL SURFACE=DIE, REF NODE=9000
```

All of the rigid surfaces in this simulation extend beyond the deformable blank to ensure that there is no possibility that slave nodes will slide behind any of them. The initial configuration of these surfaces and the locations of their reference nodes are shown in Figure 12–20.

Deformable surfaces

Using the two element sets defined on the blank, create a contact surface on the top of the blank, called **BLANK_T**, and one on the bottom, called **BLANK_B**. If you use the automatic free surface generation capability, the option blocks will look like

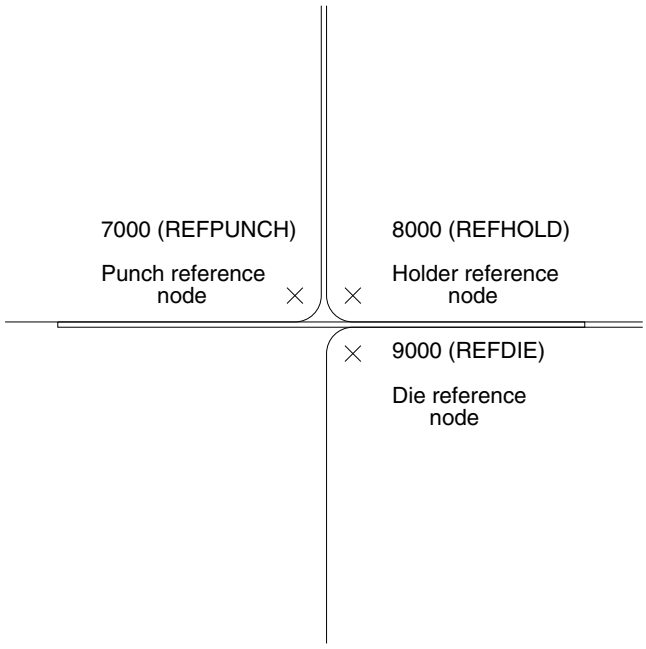


Figure 12–20 Rigid body reference nodes.

```
*SURFACE, NAME=BLANK_B
BLANK_B,
*SURFACE, NAME=BLANK_T
BLANK_T,
```

Contact pairs

Contact must be defined between the top of the blank and the punch, the top of the blank and the blank holder, and the bottom of the blank and the die. The rigid surface must be the master surface in each of these contact pairs. Each contact pair must refer to a *SURFACE INTERACTION option that defines a surface interaction model governing how the surfaces of the contact pair interact with each other. Multiple contact pairs can refer to the same *SURFACE INTERACTION option.

In this example we assume that the friction coefficient is zero between the blank and the punch. The friction coefficient between the blank and the other two tools is assumed to be 0.1. Therefore, two *SURFACE INTERACTION options must be used in the model: one with friction and one without. Frictionless contact is the default in Abaqus, so no *FRICTION option is needed in the surface interaction definition for the contact pair.

The option blocks to define the contact pairs and surface interactions in your model will look like

```
*CONTACT PAIR, INTERACTION=FRIC, TYPE=SURFACE TO SURFACE
BLANK_B, DIE
BLANK_T, HOLDER
*CONTACT PAIR, INTERACTION=NOFRIC, TYPE=SURFACE TO SURFACE
BLANK_T, PUNCH
*SURFACE INTERACTION, NAME=FRIC
*FRICTION
0.1,
*SURFACE INTERACTION, NAME=NOFRIC
```

For each contact pair the surface-to-surface contact discretization technique has been used, which controls the location where contact constraints will be generated and enforced.

12.5.6 Reviewing the input file—the history data

There are two major sources of difficulty in Abaqus/Standard contact analyses: rigid body motion of the components before contact conditions constrain them, and sudden changes in contact conditions, which lead to severe discontinuity iterations as Abaqus/Standard tries to establish the exact condition of all contact surfaces. Therefore, wherever possible, take precautions to avoid these situations.

Removing rigid body motion is not particularly difficult. Simply ensure that there are enough constraints to prevent all rigid body motions of all the components in the model. This approach may mean using boundary conditions initially to get the components into contact, instead of applying loads directly. Using this approach may require more steps than originally anticipated, but the solution of the problem should proceed more smoothly.

Alternatively, contact controls may be used to stabilize rigid body motion automatically. With this approach Abaqus/Standard applies viscous damping to the slave nodes of the contact pair. Care must be taken, however, to ensure that the viscous damping does not significantly alter the physics of the problem, as will be the case if the dissipated stabilization energy and contact damping stresses are sufficiently small.

The simulation will consist of two steps. Since the simulation involves material, geometric, and boundary nonlinearities, general steps must be used. In addition, the forming process is quasi-static; thus, we can ignore inertia effects throughout the simulation. Rather than use additional steps to establish firm contact, contact stabilization as described above will be used.

Step 1

In this step contact will be established between the blank holder and the blank while the punch and die are held fixed. Given the quasi-static nature of the problem and the fact that nonlinear response will be considered, a static, general step is required. The effects of geometric nonlinearity must be considered in this simulation, so set the NLGEOM parameter equal to YES on the *STEP option. Set the initial time increment to `0.05` and the total time period to `1.0`.

Constrain the blank holder in degrees of freedom 1 and 6, where degree of freedom 6 is the rotation in the plane of the model; constrain the punch and die completely. All of the boundary conditions for the rigid surfaces are applied to their respective rigid body reference nodes. Apply symmetric boundary constraints on the nodes of the blank lying on the symmetry plane (node set **CENTER**).

Recall that in this simulation the required blank holder force is 440 kN. Thus, apply a concentrated force to set **REFHOLD** and specify a magnitude of **-440.E3** for degree of freedom 2.

Finally, specify that the preselected field output be written every 20 increments for this step. In addition, request that the vertical reaction force and displacement (RF2 and U2) at the punch reference node (node set **REFPUNCH**) be written every increment as history data. Use the *PRINT, CONTACT=YES option to write contact diagnostics to the message file.

The complete step definition in your model appears below:

```
*STEP, NLGEOM=YES
Apply holder force
*STATIC
0.05, 1.0
*BOUNDARY
CENTER  , XSYMM
REFDIE  , 1, 6
REFPUNCH, 1, 6
REFHOLD , 1, 1
REFHOLD , 6, 6
*CLOAD
REFHOLD, 2, -4.4E5
*OUTPUT, FIELD, FREQ=20, VAR=PRESELECT
*OUTPUT, HISTORY, FREQ=1, VAR=PRESELECT
*NODE OUTPUT, NSET=REFPUNCH
RF2, U2
*PRINT, CONTACT=YES
*END STEP
```

Step 2

Move the punch down to complete the forming operation.

Between the frictional sliding, the changing contact conditions, and the inelastic material behavior, there is significant nonlinearity in this step; therefore, set the maximum number of increments to a large value (for example, set INC=1000 on the *STEP option). Set the initial time increment to be 0.05 and the total step time to be 1.0.

To alleviate convergence difficulties that may arise due to the changing contact states (in particular for contact between the punch and the blank), define contact controls to invoke automatic

contact stabilization for the contact pair involving the punch and the blank. Reduce the default damping factor by a factor of 1,000 to minimize the effects of stabilization on the solution.

Your output requests from the previous step will be propagated to this step. The input for Step 2 is

```
*STEP, NLGEOM=YES, INC=1000
Apply punch stroke
*STATIC
.05, 1.0
*CONTACT CONTROLS, MASTER=PUNCH, SLAVE=BLANK_T, STABILIZE=0.001
*BOUNDARY
REFPUNCH, 2, 2, -0.030
*END STEP
```

12.5.7 Running the analysis

Save the input in the file **channel.inp** (see "Forming a channel with Abaqus/Standard," Section A.13).

```
abaqus job=channel
```

Check the status and message files while the job is running to see how it is progressing.

Status file

This analysis should take approximately 180 increments to complete. The top of the status file is shown below.

```
 SUMMARY OF JOB INFORMATION:
STEP   INC ATT SEVERE EQUIL TOTAL   TOTAL       STEP        INC OF       DOF    IF
               DISCON ITERS ITERS   TIME/     TIME/LPF    TIME/LPF    MONITOR RIKS
               ITERS                FREQ
  1     1   1     4      0     4   0.0500      0.0500      0.05000
  1     2   1     2      0     2   0.100       0.100       0.05000
  1     3   1     2      0     2   0.175       0.175       0.07500
  1     4   1     2      0     2   0.288       0.288       0.1125
  1     5   1     3      0     3   0.456       0.456       0.1688
  1     6   1     2      0     2   0.709       0.709       0.2531
  1     7   1     2      0     2   1.00        1.00        0.2906
  2     1   1U    2      1     3   1.00        0.000       0.05000
  2     1   2U    4      0     4   1.00        0.000       0.01250
  2     1   3    29      0    29   1.00        0.00313     0.003125
  2     2   1     5      0     5   1.01        0.00547     0.002344
  2     3   1     6      0     6   1.01        0.00781     0.002344
  2     4   1     9      0     9   1.01        0.0113      0.003516
```

Abaqus has a difficult time determining the contact state in the first increment of Step 2. It needs three attempts before it finds the proper configuration of the **PUNCH** and **BLANK_T** surfaces and achieves equilibrium. After this difficult start, Abaqus quickly increases the increment size to a more reasonable value. The end of the status file is shown below.

```
 2   167   2     0     4     4  1.95        0.952       0.002239
 2   168   1     1     3     4  1.96        0.956       0.003358
 2   169   1     2     2     4  1.96        0.961       0.005037
 2   170   1     1     3     4  1.97        0.968       0.007556
 2   171   1     3     3     6  1.98        0.980       0.01133
 2   172   1U    4     0     4  1.98        0.980       0.01700
 2   172   2     1     3     4  1.98        0.984       0.004250
 2   173   1     3     2     5  1.99        0.990       0.006375
 2   174   1U    4     0     4  1.99        0.990       0.009563
 2   174   2     3     2     5  1.99        0.993       0.002391
 2   175   1     1     2     3  2.00        0.996       0.003586
 2   176   1     4     1     5  2.00        1.00        0.003721

THE ANALYSIS HAS COMPLETED SUCCESSFULLY
```

This simulation contains many severe discontinuity iterations. The message file will be quite large because of the number of iterations in the analysis. Although it might be tempting to limit the information written to this file, generally this should not be done because this information is the main source of diagnostic data that Abaqus provides during the simulation.

12.5.8 Troubleshooting Abaqus/Standard contact analyses

Contact analyses are generally more difficult to complete than just about any other type of simulation in Abaqus/Standard. Therefore, it is important to understand all of the options available to help you with contact analyses.

If a contact analysis runs into difficulty, the first thing to check is whether the contact surfaces are defined correctly. The easiest way to do this is to run a **datacheck** analysis and plot the surface normals in Abaqus/Viewer. You can plot all of the normals, for both surfaces and structural elements, on either the deformed or the undeformed plots. Use the **Normals** options in the **Common Plot Options** dialog box to do this, and confirm that the surface normals are in the correct directions.

Abaqus/Standard may still have some problems with contact simulations, even when the contact surfaces are all defined correctly. One reason for these problems may be the default convergence tolerances and limits on the number of iterations: they are quite rigorous. In contact analyses it is sometimes better to allow Abaqus/Standard to iterate a few more times rather than abandon the increment and try again. This is why Abaqus/Standard makes the distinction between severe discontinuity iterations and equilibrium iterations during the simulation.

The *PRINT, CONTACT=YES option is essential for almost every contact analysis. The information this option provides in the message file can be vital for spotting mistakes or problems. For example, chattering can be spotted because the same slave node will be seen to be involved in all of the severe discontinuity iterations. If you see this, you will have to modify the mesh in the region around that node or add constraints to the model. Contact data in the message file can also identify regions where only a single slave node is interacting with a surface. This is a very unstable situation and can cause convergence problems. Again, you should modify the model to increase the number of elements in such regions.

12.5.9 Postprocessing

In Abaqus/Viewer, examine the deformation of the blank.

Deformed model shape and contour plots

The basic result of this simulation is the deformation of the blank and the plastic strain caused by the forming process. We can plot the deformed model shape and the plastic strain, as described below.

To plot the deformed model shape:

1. Plot the deformed model shape. You can remove the die and the punch from the display and visualize just the blank.
2. In the Results Tree, expand the **Element sets** container underneath the output database file named **`channel.odb`**.
3. From the list of available element sets, select **PART-1–1.BLANK**. Click mouse button 3, and select **Replace** from the menu that appears to replace the current display group with the selected elements. Click , if necessary, to fit the model in the viewport.
   The resulting plot is shown in Figure 12–21.

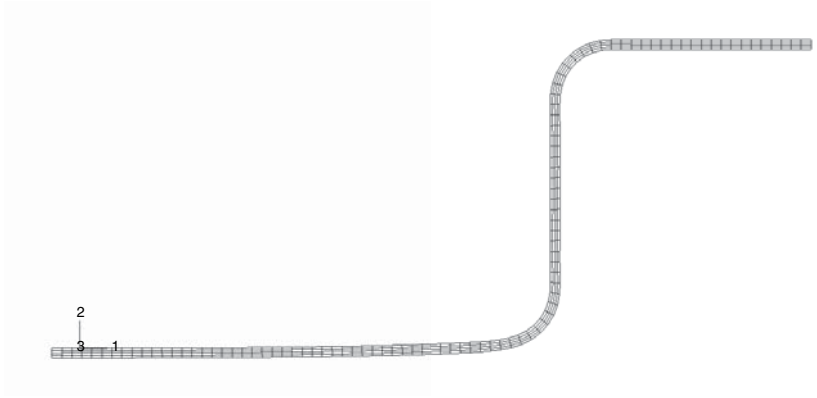


Figure 12–21 Deformed shape of blank at the end of Step 2.

To plot the contours of equivalent plastic strain:

1. From the main menu bar, select **Plot→Contours→On Deformed Shape**; or click the tool from the toolbox to display contours of Mises stress.

2. Open the **Contour Plot Options** dialog box.
3. Drag the **Contour Intervals** slider to change the number of contour intervals to **7**.
4. Click **OK** to apply these settings.
5. Select **Primary** from the list of variable types on the left side of the **Field Output** toolbar, and select **PEEQ** from the list of output variables.

   PEEQ is an integrated measure of plastic strain. A nonintegrated measure of plastic strain is PEMAG. PEEQ and PEMAG are equal for proportional loading.
6. Use the tool to zoom into any region of interest in the blank, as shown in Figure 12–22.

The maximum plastic strain is approximately 21%. Compare this with the failure strain of the material to determine if the material will tear during the forming process.

History plots of the reaction forces on the blank and punch

The solid line in Figure 12–23 shows the variation of the reaction force RF2 at the punch's rigid body reference node.

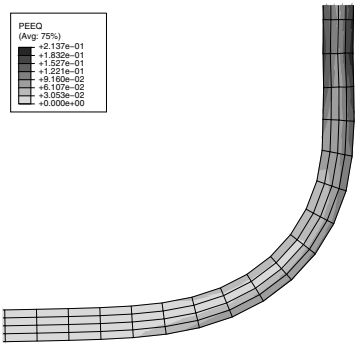


Figure 12–22 Contours of the scalar plastic strain variable PEEQ in one corner of the blank.

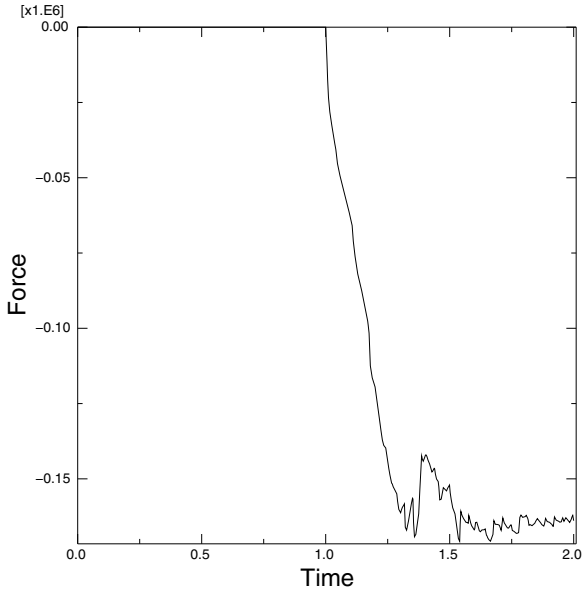


Figure 12–23 Force on punch.

To create a history plot of the reaction force:

1. In the Results Tree, expand the **History Output** container. Double-click **`Reaction force: RF1 PI: PART-1-1 Node xxx in NSET NOUT`**.

   A history plot of the reaction force in the 1-direction appears.
2. Open the **Axis Options** dialog box to label the axes.
3. Switch to the **Title** tabbed page.
4. Specify **`Reaction Force - RF2`** as the *Y*-axis label, and **`Total Time`** as the *X*-axis label.
5. Click **Dismiss** to close the dialog box.

The punch force, shown in Figure 12–23, rapidly increases to about 160 kN during Step 2, which runs from a total time of 1.0 to 2.0.

History plot of the stabilization and internal energies

It is important to verify that the presence of contact stabilization does not significantly alter the physics of the problem. One way to assess this requirement is to compare the energy dissipation due to stabilization (ALLSD) against the internal energy of the structure (ALLIE). Ideally the amount of stabilization energy should be a small fraction of the internal energy. Figure 12–24 shows the variation of the stabilization and internal energies. It is clear that the dissipated stabilization energy is indeed small.

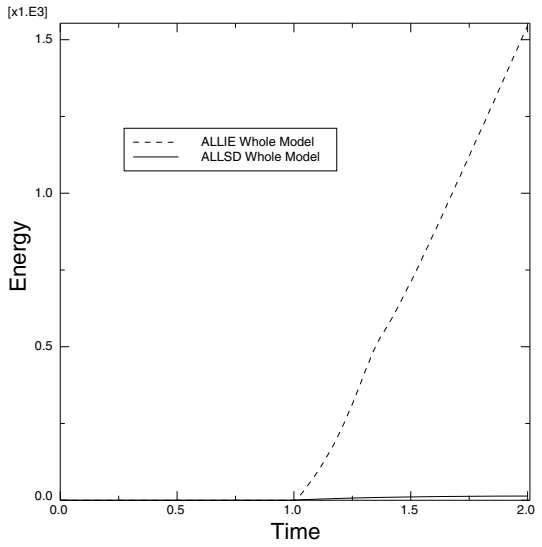


Figure 12–24 Stabilization and internal energies.

Plotting contours on surfaces

Abaqus/Viewer includes a number of features designed specifically for postprocessing contact analyses. The **Display Group** feature can be used to collect surfaces into display groups, similar to element and node sets.

To display contact surface normal vectors:

1. Plot the undeformed model shape.
2. In the Results Tree, expand the **Surface Sets** container. Select the surfaces named **`BLANK-T`** and **`PART-1-1.PUNCH`**. Click mouse button 3, and select **Replace** from the menu that appears.
3. Using the **Common Plot Options** dialog box, turn on the display of the normal vectors (**On surfaces**) and set the length of the vector arrows to **Short**.
4. Use the tool, if necessary, to zoom into any region of interest, as shown in Figure 12–25.

To contour the contact pressure:

1. Plot the contours of plastic strain again.
2. From the list of variable types on the left side of the **Field Output** toolbar, select **Primary**, if it is not already selected.
3. From the list of output variables in the center of the toolbar, select **`CPRESS`**.
4. Remove the **`PART-1-1.PUNCH`** surface from your display group.

   To visualize contours of surface-based variables better in two-dimensional models, you can extrude the plane strain elements to construct the equivalent three-dimensional view. You can sweep axisymmetric elements in a similar fashion.
5. From the main menu bar, select **View→ODB Display Options**.

   The **ODB Display Options** dialog box appears.
6. Select the **Sweep/Extrude** tab to access the **Sweep/Extrude** options.
7. In the **Extrude** region of the dialog box, toggle on **Extrude elements**; and set the **Depth** to **`0.05`** to extrude the model for the purpose of displaying contours.
8. Click **OK** to apply these settings.

   Rotate the model using the tool to display the model from a suitable view, such as the one shown in Figure 12–26.

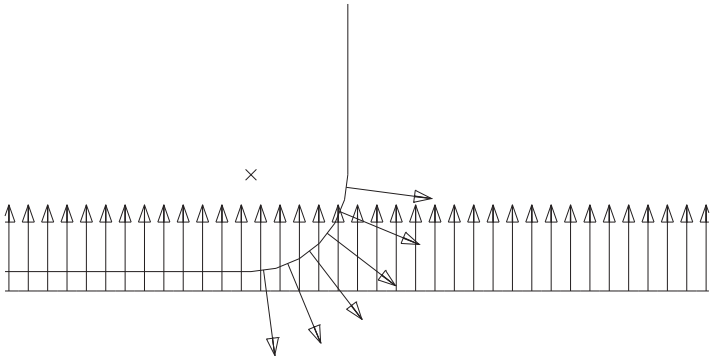


Figure 12–25 Surface normals.

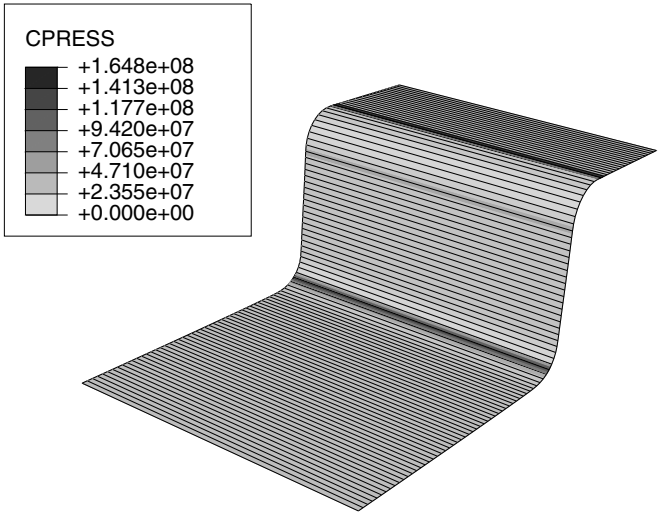


Figure 12–26 Contact pressure.

12.6 General contact in Abaqus/Standard

In the channel forming example in "Abaqus/Standard 2-D example: forming a channel," Section 12.5, contact interaction is defined using the contact pairs algorithm, which requires you to explicitly define the surfaces that may potentially come into contact. As an alternative, you can specify contact in an Abaqus/Standard analysis by using the general contact algorithm. The contact interaction domain, contact properties, and surface attributes are specified independently for general contact, offering a more flexible way to add detail incrementally to a model. The simple interface for specifying general contact allows for a highly automated contact definition; however, it is also possible to define contact with the general contact interface to mimic traditional contact pairs. Conversely, specifying self-contact of a surface spanning multiple bodies with the contact pair user interface (if the surface-to-surface formulation is used) mimics the highly automated approach often used for general contact.

In Abaqus/Standard, traditional pairwise specifications of contact interactions will often result in more efficient analyses as compared to an all-inclusive self-contact approach to defining contact. Therefore, there is often a trade-off between ease of defining contact and analysis performance. Abaqus/CAE provides a contact detection tool that greatly simplifies the process of creating traditional contact pairs for Abaqus/Standard (see "Understanding contact and constraint detection," Section 15.6 of the Abaqus/CAE User's Manual).

12.7 Abaqus/Standard 3-D example: shearing of a lap joint

This simulation of the shearing of a lap joint illustrates the use of general contact in Abaqus/Standard.

The model consists of two overlapping aluminum plates that are connected with a titanium rivet. The left end of the bottom plate is fixed, and the force is applied to the right end of the top plate to shear the joint. Figure 12–27 shows the basic arrangement of the components. Because of symmetry, only half of the joint is modeled to reduce computational cost. Frictional contact is assumed.

12.7.1 Mesh design

Select the element type before designing the mesh. The mesh used for the plates should consist of C3D8I elements; the rivet should be meshed with C3D8R and C3D6 elements (a representative mesh is shown in Figure 12–28).

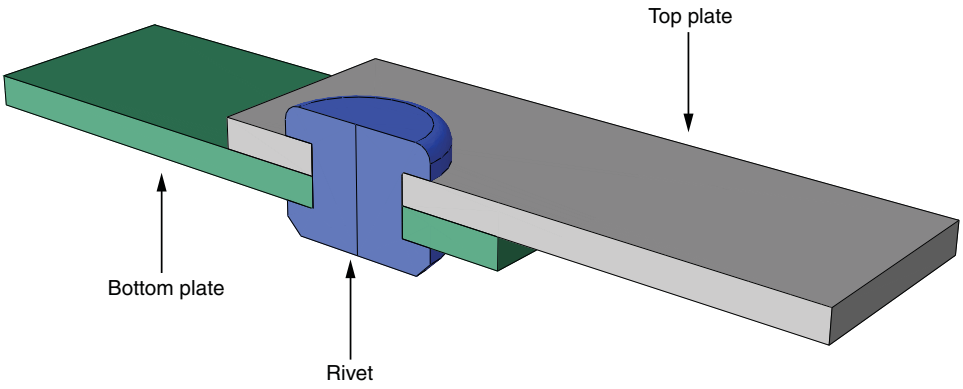


Figure 12–27 Lap joint analysis.

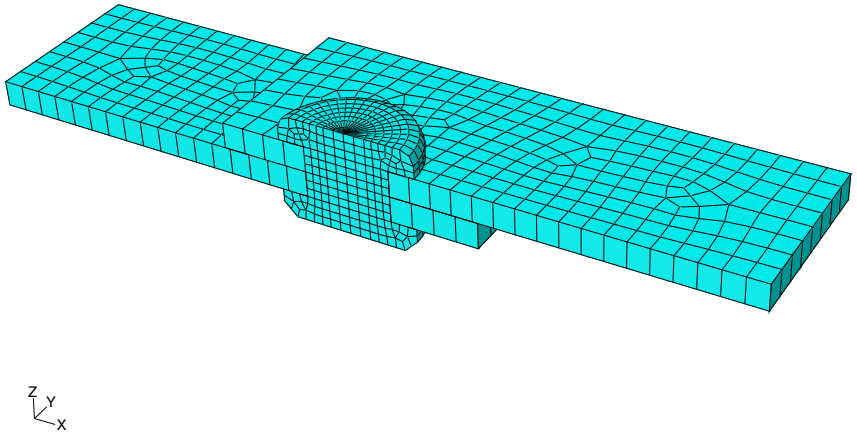


Figure 12–28 Mesh.

12.7.2 Preprocessing—creating the model

The steps that follow assume that you have access to the full input file for this example. This input file, **`lap_joint.inp`**, is provided in "Shearing of a lap joint," Section A.14, in the online HTML version of this manual. Instructions on how to fetch and run the script are given in Appendix A, "Example Files." If you wish to create the entire model using Abaqus/CAE, please refer to "Abaqus/Standard 3-D example: shearing of a lap joint," Section 12.8 of Getting Started with Abaqus: Interactive Edition.

12.7.3 Reviewing the input file—the model data

We first review the model definition, including the node and element definitions, and section and material properties.

Model description

The input file starts with a relevant description of the simulation and model in the *HEADING option.

```
*HEADING
Shearing of a lap joint
SI units (N, kg, mm, s)
```

Nodal coordinates and element connectivity

Check that the preprocessor used the correct element type for the plates and rivet. Provide meaningful element set names, such as **PLATE** and **RIVET**, for the elements. The *ELEMENT options in this model follow:

```
*ELEMENT, TYPE=C3D8I, ELSET=PLATE
...
*ELEMENT, TYPE=C3D8R, ELSET=RIVET
...
*ELEMENT, TYPE=C3D6,  ELSET=RIVET
...
```

The model definition also specifies the creation of node sets so that parts of the model can be constrained and moved easily. These sets are located at the following locations: at the bottom-left corner of the bottom plate (set **CORNER**), the left face of the bottom plate (set **FIX**), the right face of the top plate (set **PULL**), and the symmetry plane (set **SYMM**).

```
*NSET, NSET=CORNER
...
*NSET, NSET=FIX
...
*NSET, NSET=PULL
...
*NSET, NSET=SYMM
...
```

The first of these sets will be used to prevent rigid body motion in the 3-direction; the next two will be used to fix the end of one plate and pull the end of the other, respectively; the last one will be used to impose symmetry conditions.

Section and material properties

The plates are made from aluminum (elastic modulus of 71.7×10^3 MPa, $\nu = 0.33$). Its stress-strain behavior is shown in Figure 12–29.

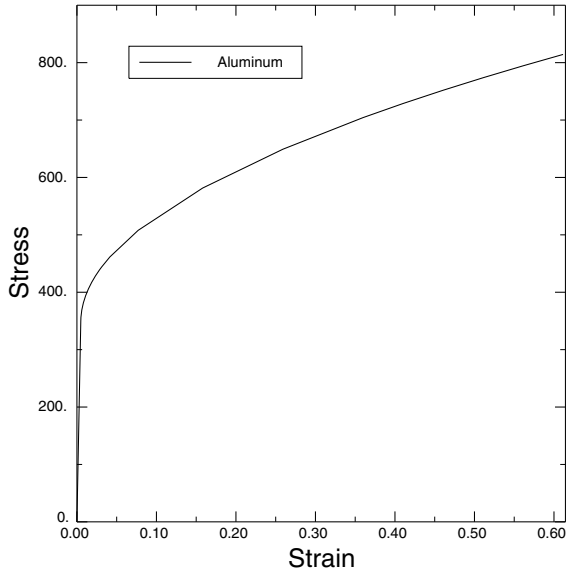


Figure 12–29 Aluminum stress-strain curve.

The rivet is made from titanium (elastic modulus of 112×10^3 MPa, $\nu = 0.34$). Its stress-strain behavior is shown in Figure 12–30.

The following input options are needed to define the material properties:

```
*SOLID SECTION, MATERIAL=ALUMINUM, ELSET=PLATES
*MATERIAL, NAME=ALUMINUM
*ELASTIC
 71700., 0.33
*PLASTIC
 350.00, 0.
 368.71, 0.001
 376.50, 0.002
 391.98, 0.005
 403.15, 0.008
 412.36, 0.011
 422.87, 0.015
 444.17, 0.025
 461.50, 0.035
 507.90, 0.070
```

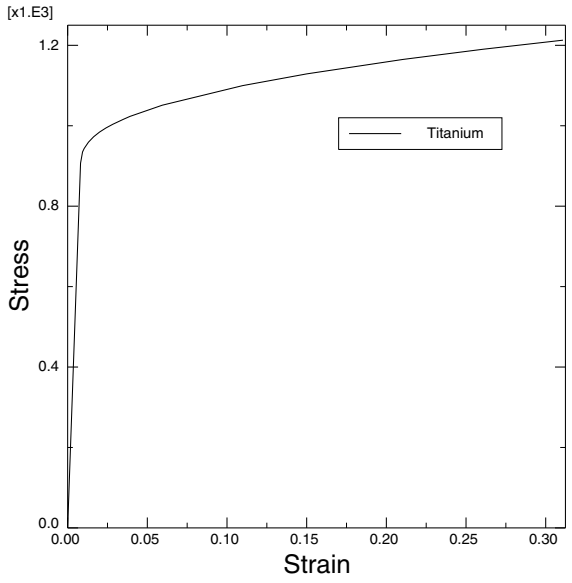


Figure 12–30 Titanium stress-strain curve.

```
 581.50, 0.150
 649.17, 0.250
 704.22, 0.350
 728.78, 0.400
 751.85, 0.450
 773.68, 0.500
 794.44, 0.550
 814.28, 0.600
*SOLID SECTION, MATERIAL=TITANIUM, ELSET=RIVET
*MATERIAL, NAME=TITANIUM
*ELASTIC
 112000., 0.34
*PLASTIC
 907.00, 0.
 934.86, 0.001
 944.28, 0.002
 961.77, 0.005
 973.73, 0.008
 983.28, 0.011
```

```
993.89, 0.015
1014.7, 0.025
1023.3, 0.030
1051.1, 0.050
1099.8, 0.100
1129.0, 0.140
1164.9, 0.200
1190.2, 0.250
1212.8, 0.300
```

12.7.4 Contact definitions

The contact definitions for the model are discussed here.

Defining contact

Contact will be used to enforce the interactions between the plates and the rivet. The friction coefficient between all parts is assumed to be 0.05.

This problem could use either contact pairs or the general contact algorithm. We will use general contact in this problem to demonstrate the simplicity of the contact definition.

The contact property is defined using the *SURFACE INTERACTION option; a friction coefficient of 0.05 is specified.

```
*SURFACE INTERACTION, NAME=FRIC
*FRICTION
 0.05,
```

Use the *CONTACT option to define a general contact interaction. Use the ALL EXTERIOR parameter on the *CONTACT INCLUSIONS option to specify self-contact for the unnamed, all-inclusive surface defined automatically by Abaqus/Standard. The *CONTACT PROPERTY ASSIGNMENT option is used to assign the contact property named **FRIC** to the general contact interaction.

```
*CONTACT
*CONTACT INCLUSIONS, ALL EXTERIOR
*CONTACT PROPERTY ASSIGNMENT
 , , FRIC
```

12.7.5 Reviewing the input file—the history data

Step definition and boundary conditions

Create a single static, general step and include the effects of geometric nonlinearity. Set the initial time increment to **0.05** and the total time to **1.0**. Accept the default output requests.

One end of the assembly is fixed while the other is pulled along the length of the plates (1-direction). In addition, a single node is fixed in the vertical (3-) direction to prevent rigid body motion and the nodes on the symmetry plane are fixed in the direction normal to the plane (2-direction). The boundary conditions are summarized in Table 12–1.

Table 12–1 Summary of boundary conditions.

Geometry Set	BCs
fix	U1 = 0.0
pull	U1 = 2.5
symm	U2 = 0.0
corner	U3 = 0.0

The complete step definition required for the model appears below:

```
*STEP, NLGEOM=YES
*STATIC
 0.05, 1.
*BOUNDARY
 FIX,    1, 1
 PULL,   1, 1, 2.5
 SYMM,   2, 2
 CORNER, 3, 3
*OUTPUT, FIELD, VARIABLE=PRESELECT
*OUTPUT, HISTORY, VARIABLE=PRESELECT
*END STEP
```

12.7.6 Running the analysis

Save the input in the file **lap_joint.inp** (see "Shearing of a lap joint," Section A.14). Run the simulation using the following command:

```
abaqus job=lap_joint
```

Check the status and message files while the job is running to see how it is progressing.

Status file

This analysis should take approximately 13 increments to complete. The contents of the status file are shown below:

```
 SUMMARY OF JOB INFORMATION:
STEP   INC ATT SEVERE EQUIL TOTAL  TOTAL      STEP       INC OF       DOF    IF
               DISCON ITERS ITERS  TIME/    TIME/LPF    TIME/LPF    MONITOR RIKS
               ITERS               FREQ
  1     1   1    11     2    13  0.0500      0.0500      0.05000
  1     2   1     3     2     5  0.100       0.100       0.05000
  1     3   1     4     2     6  0.175       0.175       0.07500
  1     4   1     3     2     5  0.288       0.288       0.1125
  1     5   1     4     5     9  0.456       0.456       0.1688
  1     6   1U    6     0     6  0.456       0.456       0.1688
  1     6   2     3     3     6  0.498       0.498       0.04219
  1     7   1     2     6     8  0.541       0.541       0.04219
  1     8   1     1     4     5  0.583       0.583       0.04219
  1     9   1     0     4     4  0.625       0.625       0.04219
  1    10   1     1     2     3  0.688       0.688       0.06328
  1    11   1     1     3     4  0.783       0.783       0.09492
  1    12   1     2     2     4  0.926       0.926       0.1424
  1    13   1     1     2     3  1.00        1.00        0.07441
```

12.7.7 Postprocessing

In Abaqus/Viewer, examine the deformation of the assembly.

Deformed model shape and contour plots

The basic results of this simulation are the deformation of the joint and the stresses caused by the shearing process. Plot the deformed model shape and the Mises stress, as shown in Figure 12–31 and Figure 12–32, respectively.

Contact pressures

You will now plot the contact pressures in the lap joint.

Since it is difficult to see contact pressures when the entire model is displayed, use the **Display Groups** toolbar to display only the top plate in the viewport.

Create a path plot to examine the variation of the contact pressure around the bolt hole of the top plate.

To create a path plot:

1. In the Results Tree, double-click **Paths**. In the **Create Path** dialog box, select **Edge list** as the type and click **Continue**.
2. In the **Edit Edge List Path** dialog box, select the instance corresponding to the top plate and click **Add After**.
3. In the prompt area, select **by shortest distance** as the selection method.

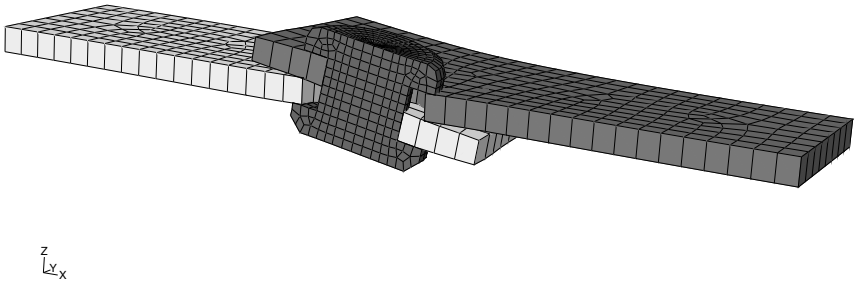


Figure 12–31 Deformed model shape.

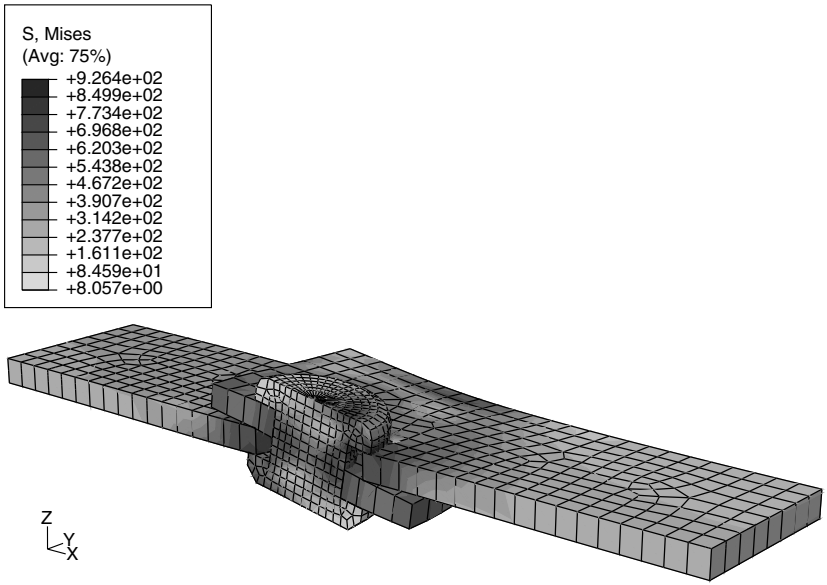


Figure 12–32 Mises stress.

4. In the viewport, select the edge at the left end of the bolt hole as the starting edge of the path and the node at the right end of the bolt hole as the end node of the path, as shown in Figure 12–33.

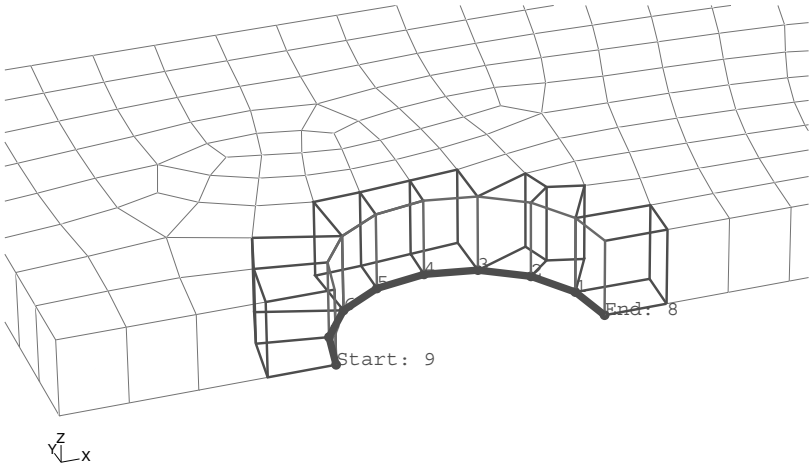


Figure 12–33 Path definition.

5. Click **Done** in the prompt area to indicate that you have finished making selections for the path. Click **OK** to save the path definition and to close the **Edit Edge List Path** dialog box.
6. In the Results Tree, double-click **XYData**. Select **Path** in the **Create XY Data** dialog box, and click **Continue**.
7. In the **Y Values** frame of the **XY Data from Path** dialog box, click **Step/Frame**. In the **Step/Frame** dialog box, select the last frame of the step. Click **OK** to close the **Step/Frame** dialog box.
8. Make sure that the field output variable is set to **CPRESS**, and click **Plot** to view the path plot. Click **Save As** to save the plot.

   The path plot appears as shown in Figure 12–34.

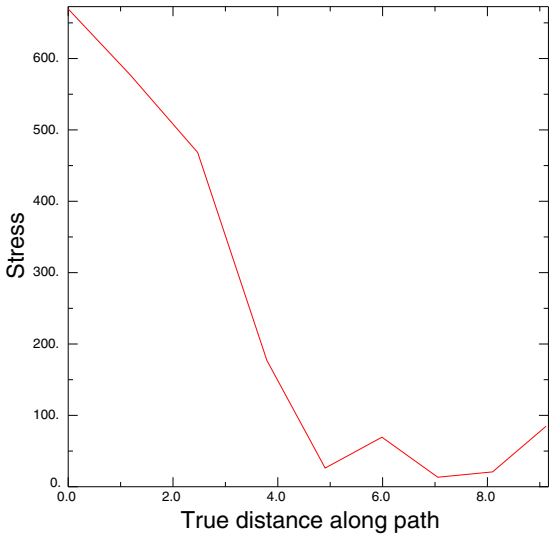


Figure 12–34 CPRESS distribution around the bolt hole in top plate.

12.8 Defining contact in Abaqus/Explicit

Abaqus/Explicit provides two algorithms for modeling contact interactions. The general (“automatic”) contact algorithm allows very simple definitions of contact with very few restrictions on the types of surfaces involved (see “Defining general contact interactions in Abaqus/Explicit,” Section 35.4.1 of the Abaqus Analysis User’s Manual). The contact pair algorithm has more restrictions on the types of surfaces involved and often requires more careful definition of contact; however, it allows for some interaction behaviors that currently are not available with the general contact algorithm (see “Defining contact pairs in Abaqus/Explicit,” Section 35.5.1 of the Abaqus Analysis User’s Manual). General contact interactions typically are defined by specifying self-contact for a default, element-based surface defined automatically by Abaqus/Explicit that includes all bodies in the model. To refine the contact domain, you can include or exclude specific surface pairs. Contact pair interactions are defined by specifying each of the individual surface pairs that can interact with each other.

12.8.1 Abaqus/Explicit contact formulation

The contact formulation in Abaqus/Explicit includes the constraint enforcement method, the contact surface weighting, and the sliding formulation.

Constraint enforcement method

For general contact (∗CONTACT) Abaqus/Explicit enforces contact constraints using a penalty contact method, which searches for node-into-face and edge-into-edge penetrations in the current configuration. The penalty stiffness that relates the contact force to the penetration distance is chosen automatically by Abaqus/Explicit so that the effect on the time increment is minimal yet the penetration is not significant.

The contact pair algorithm (∗CONTACT PAIR) uses a kinematic contact formulation by default that achieves precise compliance with the contact conditions using a predictor/corrector method. The increment at first proceeds under the assumption that contact does not occur. If at the end of the increment there is an overclosure, the acceleration is modified to obtain a corrected configuration in which the contact constraints are enforced. The predictor/corrector method used for kinematic contact is discussed in more detail in "Contact constraint enforcement methods in Abaqus/Explicit," Section 37.2.3 of the Abaqus Analysis User's Manual; some limitations of this method are discussed in "Common difficulties associated with contact modeling using contact pairs in Abaqus/Explicit," Section 38.2.2 of the Abaqus Analysis User's Manual.

The normal contact constraint for contact pairs can optionally be enforced with the penalty contact method, which can model some types of contact that the kinematic method cannot. For example, the penalty method allows modeling of contact between two rigid surfaces (except when both surfaces are analytical rigid surfaces). When the penalty contact formulation is used, equal and opposite contact forces with magnitudes equal to the penalty stiffness times the penetration distance are applied to the master and slave nodes at the penetration points. The penalty stiffness is chosen automatically by Abaqus/Explicit and is similar to that used by the general contact algorithm. A penalty scale factor can also be specified. To select the penalty method for a contact pair analysis, set the MECHANICAL CONSTRAINT parameter to PENALTY on the ∗CONTACT PAIR option.

Contact surface weighting

In the pure master-slave approach one of the surfaces is the master surface and the other is the slave surface. As the two bodies come into contact, the penetrations are detected and the contact constraints are applied according to the constraint enforcement method (kinematic or penalty). Pure master-slave weighting (regardless of the constraint enforcement method) will resist only penetrations of slave nodes into master facets. Penetrations of master nodes into the slave surface can go undetected, as shown in Figure 12–35, unless the mesh on the slave surface is adequately refined.

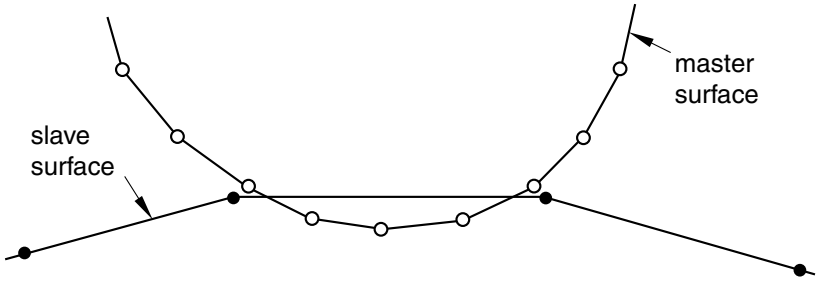


Figure 12–35 Penetration of master nodes into slave surface with pure master-slave contact.

Balanced master-slave contact simply applies the pure master-slave approach twice, reversing the surfaces on the second pass. One set of contact constraints is obtained with surface 1 as the slave, and another set of constraints is obtained with surface 2 as the slave. The acceleration corrections or forces are obtained by taking a weighted average of the two calculations. For kinematic balanced master-slave contact a second correction is made to resolve any remaining penetrations, as described in "Contact formulations for contact pairs in Abaqus/Explicit," Section 37.2.2 of the Abaqus Analysis User's Manual. The balanced master-slave contact constraint when kinematic compliance is used is illustrated in Figure 12–36.

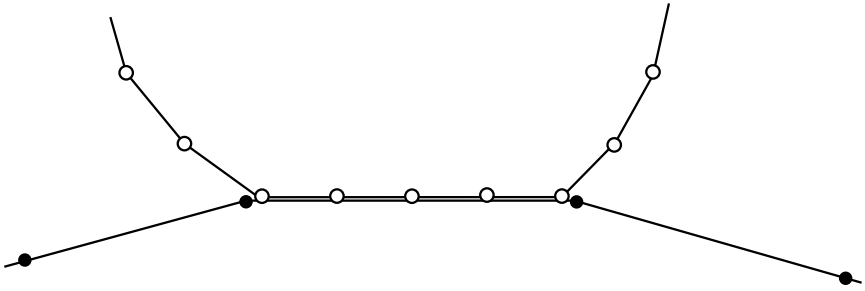


Figure 12–36 Balanced master-slave contact constraint with kinematic compliance.

The balanced approach minimizes the penetration of the contacting bodies and, thus, provides more accurate results in most cases.

The general contact algorithm uses balanced master-slave weighting whenever possible; pure master-slave weighting is used for general contact interactions involving node-based surfaces, which can act only as pure slave surfaces. For the contact pair algorithm Abaqus/Explicit will decide which type of weighting to use for a given contact pair based on the nature of the two surfaces involved and the constraint enforcement method used.

The general contact algorithm uses balanced master-slave weighting whenever possible; pure master-slave weighting is used for general contact interactions involving node-based surfaces,

which can act only as pure slave surfaces. Use the *CONTACT FORMULATION, TYPE=PURE MASTER-SLAVE to specify pure master-slave weighting for other general contact interactions.

For the contact pair algorithm Abaqus/Explicit will decide which type of weighting to use for a given contact pair based on the nature of the two surfaces involved and the constraint enforcement method used. The weight of the average can be specified by the user for balanced master-slave contact with the contact pair algorithm using the WEIGHT parameter on the *CONTACT PAIR option. For most element types the default weight is 0.5 so that the same weight is used for each of the acceleration corrections. Setting WEIGHT to 1.0 specifies a pure master-slave relationship with the first surface as the master surface. Conversely, a weight of zero means that the second surface is the master surface.

Sliding formulation

When defining a contact pair, you must decide whether the magnitude of the relative sliding will be small or finite. The default (and only option for general contact interactions) is the more general finite-sliding formulation. Small sliding is appropriate if the relative motion of the two surfaces is less than a small proportion of the characteristic length of an element face. The small-sliding formulation is selected by including the SMALL SLIDING parameter on the *CONTACT PAIR option. Using the small-sliding formulation when applicable results in a more efficient analysis.

12.9 Modeling considerations in Abaqus/Explicit

We now discuss the following modeling considerations: correct definition of surfaces, overconstraints, mesh refinement, and initial overclosures.

12.9.1 Correct surface definitions

Certain rules must be followed when defining surfaces for use with each of the contact algorithms. The general contact algorithm has fewer restrictions on the types of surfaces that can be involved in contact; however, two-dimensional and node-based surfaces can be used only with the contact pair algorithm.

Continuous surfaces

Surfaces used with the general contact algorithm can span multiple unattached bodies. More than two surface facets can share a common edge. In contrast, all surfaces used with the contact pair algorithm must be continuous and simply connected. The continuity requirement has the following implications for what constitutes a valid or invalid surface definition for the contact pair algorithm:

- In two dimensions the surface must be either a simple, nonintersecting curve with two terminal ends or a closed loop. Figure 12–37 shows examples of valid and invalid two-dimensional surfaces.

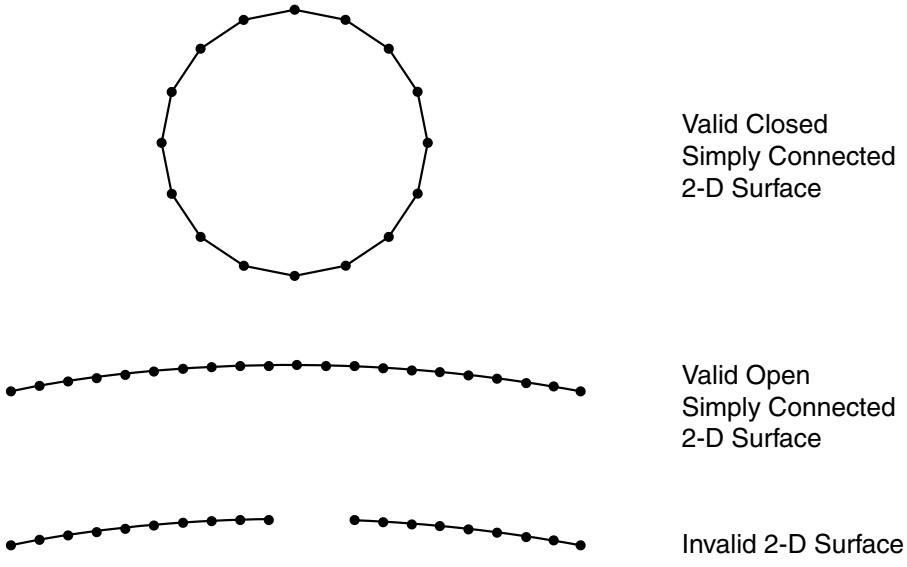
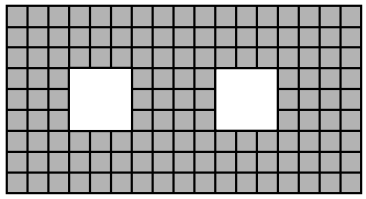


Figure 12–37 Valid and invalid two-dimensional surfaces for the contact pair algorithm.

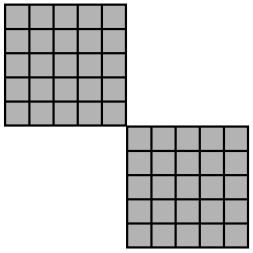
- In three dimensions an edge of an element face belonging to a valid surface may be either on the perimeter of the surface or shared by one other face. Two element faces forming a contact surface cannot be joined just at a shared node; they must be joined across a common element edge. An element edge cannot be shared by more than two surface facets. Figure 12–38 illustrates valid and invalid three-dimensional surfaces.
- In addition, it is possible to define three-dimensional, double-sided surfaces. In this case both sides of a shell, membrane, or rigid element are included in the same surface definition, as shown in Figure 12–39.

Extending surfaces

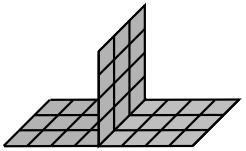
Abaqus/Explicit does not extend surfaces automatically beyond the perimeter of the surface defined by the user. If a node from one surface is in contact with another surface and it slides along the surface until it reaches an edge, it may "fall off the edge." Such behavior can be particularly troublesome because the node may later reenter from the back side of the surface, thereby violating the kinematic constraint and causing large jumps in acceleration at that node. Consequently, it is good modeling practice to extend surfaces somewhat beyond the regions that will actually contact. In general, we recommend covering each contacting body entirely with surfaces; the additional computational expense is minimal.



valid simply connected surface

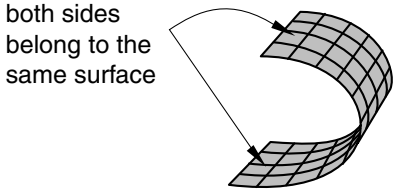


invalid surface



invalid surface

Figure 12–38 Valid and invalid three-dimensional surfaces for the contact pair algorithm.



valid double-sided surface

Figure 12–39 Valid double-sided surface.

Figure 12–40 shows two simple box-like bodies constructed of brick elements. The upper box has a contact surface defined only on the top face of the box. While it is a permissible surface definition in Abaqus/Explicit, the lack of extensions beyond the “raw edge” could be problematic. In the lower box the surface wraps some distance around the side walls, thereby extending beyond the flat, upper surface. If contact is to occur only at the top face of the box, this extended surface definition minimizes contact problems by keeping any contacting nodes from going behind the contact surface.

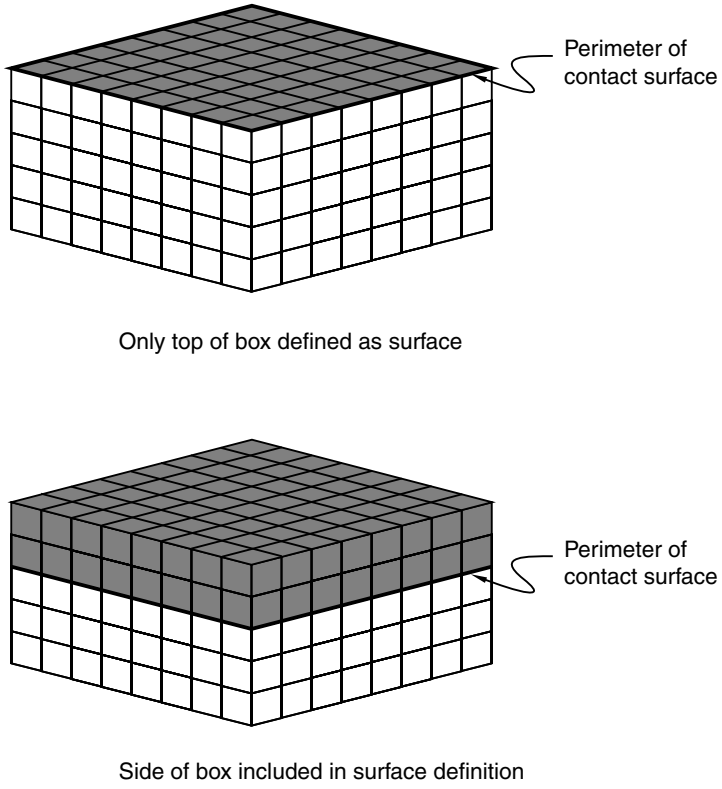


Figure 12–40 Surface perimeters.

Mesh seams

Two nodes with the same coordinate (double nodes) can generate a seam or crack in a valid surface that appears to be continuous, as shown in Figure 12–41. A node sliding along the surface can fall through this crack and slide behind the contact surface. A large, nonphysical acceleration correction may be caused once penetration is detected. Mesh seams can be detected in Abaqus/Viewer by drawing the free edges of the model. Any seams that are not part of the desired perimeter can be double-noded regions.

Complete surface definition

Figure 12–42 illustrates a two-dimensional model of a simple connection between two parts.

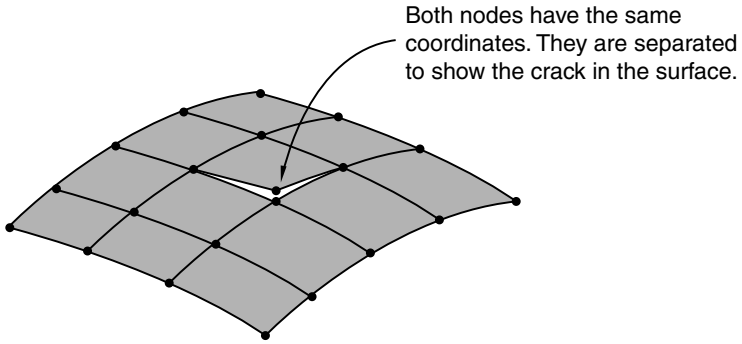


Figure 12–41 Example of a double-noded mesh.

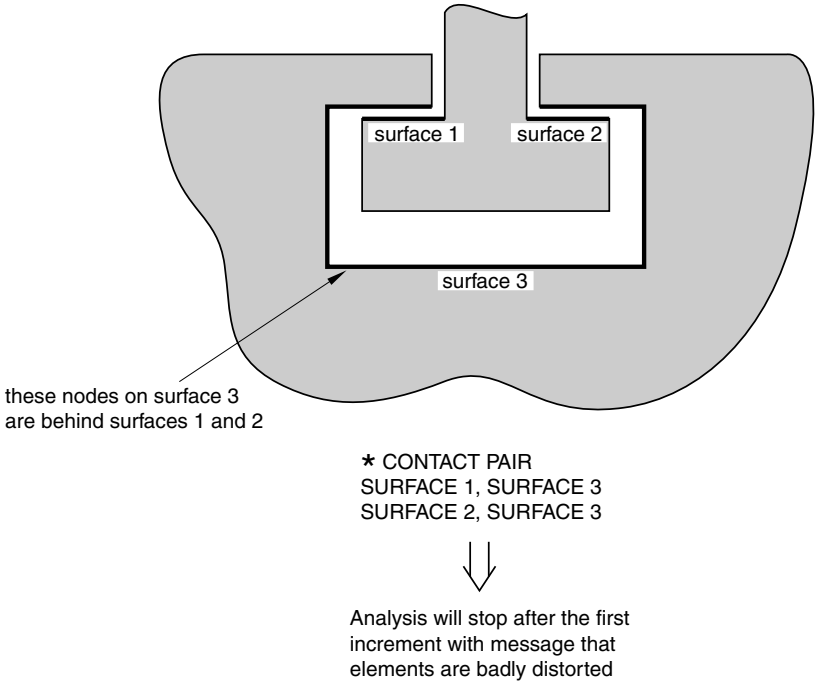


Figure 12–42 Example of an incorrect surface definition.

The contact definition shown in the figure is not adequate for modeling this connection because the surfaces do not represent a complete description of the geometry of the bodies. At the beginning of

the analysis some of the nodes on surface 3 find that they are "behind" surfaces 1 and 2. Figure 12–43 shows an adequate surface definition for this connection. The surfaces are continuous and describe the entire geometry of the contacting bodies.

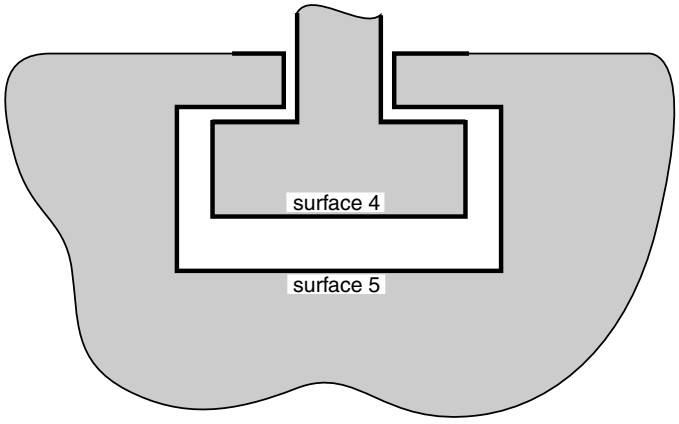
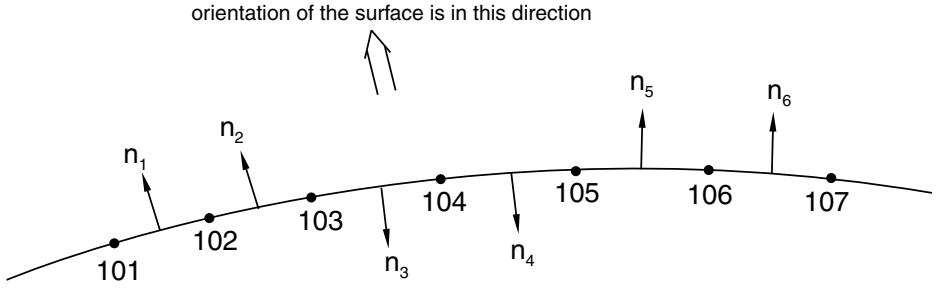


Figure 12–43 Correct surface definition.

Consistent surface normals

Single-sided surfaces on shell, membrane, or rigid elements must be defined so that the normal directions do not "flip" as the surface is traversed. Figure 12–44 shows a mesh of SAX1 elements whose normals are not continuous from one element to the next. The face identifier SPOS indicates that the surface is on the face with the positive outward normal, and the face identifier SNEG indicates the reverse. If a surface was defined using the SPOS face for all the elements, Abaqus/Explicit would issue a warning message stating that the surface is not valid. A valid surface could be defined with this mesh if the surface definition shown in the figure, which uses both SPOS and SNEG face identifiers to accommodate the inconsistent element normals, is used.

It is not necessary for the normals of all the underlying shell, membrane, or rigid elements to have a consistent positive orientation for a double-sided surface; if possible, Abaqus/Explicit will define the surface such that its facets have consistent normals, even if the underlying elements do not have consistent normals. The facet normals will be the same as the element normals if the element normals are all consistent; otherwise, an arbitrary positive orientation is chosen for the surface. If it is not possible to make the facet normals consistent (for example, if the surface contains a T-intersection of shells), the surface can be used with the general contact algorithm but not with the contact pair algorithm.




```
.
.
*ELEMENT, TYPE = SAX1, ELSET = SHELL
 1, 101, 102
 2, 102, 103
 3, 104, 103
 4, 105, 104
 5, 105, 106
 6, 106, 107
   .
   .
   .
*SURFACE DEFINITION, NAME = WALL
 1, SPOS
 2, SPOS
 3, SNEG
 4, SNEG
 5, SPOS
 6, SPOS
```

Figure 12–44 Inconsistent surface normals.

Highly warped surfaces

No special treatment of warped surfaces is required for the general contact algorithm. However, when a surface used with the contact pair algorithm contains highly warped facets, a more expensive tracking approach must be used than the approach required when the surface does not contain highly warped facets. To keep the solution as efficient as possible, Abaqus monitors the warpage of the surfaces and issues a warning if surfaces become too warped; if the normal directions of adjacent facets differ by more than 20°, Abaqus issues a warning message. Once a surface is deemed to be highly warped, Abaqus switches from the more efficient contact search approach to a more accurate search approach to account for the difficulties posed by the highly warped surface.

For the sake of efficiency Abaqus does not check for highly warped surfaces every increment. Rigid surfaces are checked for high warpage only at the start of the step, since rigid surfaces do not change shape during the analysis. Deformable surfaces are checked for high warpage every 20 increments by default. Some analyses may have surfaces whose warpage increases in severity quite suddenly, making the default 20 increment frequency check inadequate. The user can change

the frequency of the warping checks by setting the WARP CHECK PERIOD parameter on the *CONTACT CONTROLS option to the desired number of increments. Some analyses in which the surface warping is less than 20° may also require the more accurate contact search approach associated with highly warped surfaces. Use the WARP CUT OFF parameter on the *CONTACT CONTROLS option to redefine the angle that defines high warpage.

Rigid element discretization

Complex rigid surface geometries can be modeled using rigid elements. Rigid elements in Abaqus/Explicit are not smoothed; they remain faceted exactly as defined by the user. The advantage of unsmoothed surfaces is that the surface defined by the user is exactly the same as the surface used by Abaqus; the disadvantage is that faceted surfaces require much higher mesh refinement to define smooth bodies accurately. In general, using a large number of rigid elements to define a rigid surface does not increase the CPU costs significantly. However, a large number of rigid elements does increase the memory overhead significantly.

The user must ensure that the discretization of any curved geometry on rigid bodies is adequate. If the rigid body discretization is too coarse, contacting nodes on the deformable body may "snag," leading to erroneous results, as illustrated in Figure 12–45.

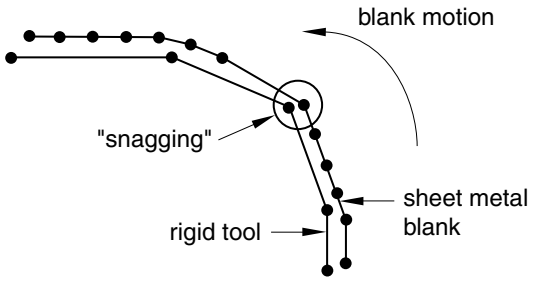


Figure 12–45 Potential effect of coarse rigid body discretization.

A node that is snagged on a sharp corner may be trapped from further sliding along the rigid surface for some time. Once enough energy is released to slide beyond the sharp corner, the node will snap dynamically before contacting the adjacent facet. Such motions cause noisy solutions. The more refined the rigid surface, the smoother the motion of the contacting slave nodes. The general contact algorithm includes some numerical rounding of features that prevents snagging of nodes from becoming a concern for discrete rigid surfaces. In addition, penalty enforcement of the contact constraints reduces the tendency for snagging to occur. Analytical rigid surfaces should normally be used with the contact pair algorithm for rigid bodies whose shape is an extruded profile or a surface of revolution.

12.9.2 Overconstraining the model

Just as multiple conflicting boundary conditions should not be defined at a given node, multi-point constraints and contact conditions enforced with the kinematic method generally should not be defined at the same node because they may generate conflicting kinematic constraints. Unless the constraints are entirely orthogonal to one another, the model will be overconstrained; the resulting solution will be quite noisy, as Abaqus/Explicit tries to satisfy the conflicting constraints. Penalty contact constraints and multi-point constraints acting on the same nodes will not generate conflicts because the penalty constraints are not enforced as strictly as the multi-point constraints.

12.9.3 Mesh refinement

For contact as well as all other types of analyses, the solution improves as the mesh is refined. For contact analyses using a pure master-slave approach, it is especially important that the slave surface is refined adequately so that the master surface facets do not overly penetrate the slave surface. The balanced master-slave approach does not require high mesh refinement on the slave surface to have adequate contact compliance. Mesh refinement is generally most important with pure master-slave contact between deformable and rigid bodies; in this case the deformable body is always the pure slave surface and, thus, must be refined enough to interact with any feature on the rigid body. Figure 12–46 shows an example of the penetration that can occur if the discretization of the slave surface is poor compared to the dimensions of the features on the master surface. If the deformable surface were more refined, the penetrations of the rigid surface would be much less severe.

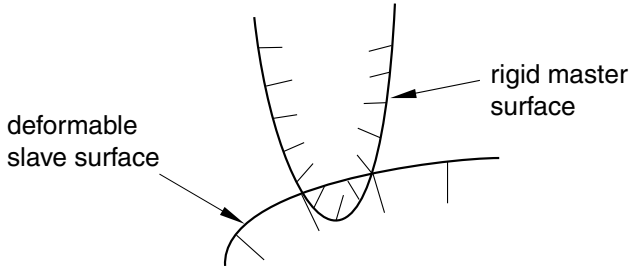


Figure 12–46 Example of inadequate slave surface discretization.

Tie constraints

Using the *TIE option prevents surfaces initially in contact from penetrating, separating, or sliding relative to one another. Tie constraints are, therefore, an easy means of mesh refinement. Since any gaps that exist between the two contact surfaces, however small, will result in nodes that are not

tied to the opposite contact boundary, you must use the ADJUST=YES parameter to ensure that the two surfaces are exactly in contact at the start of the analysis.

The tie constraint formulation constrains translational and, optionally, rotational degrees of freedom. When using tied contact with structural elements, you must ensure that any unconstrained rotations will not cause problems.

12.9.4 Initial contact overclosure

Abaqus/Explicit will automatically adjust the undeformed coordinates of nodes on contact surfaces to remove any initial overclosures. When using the balanced master-slave approach, both surfaces are adjusted; when using the pure master-slave approach, only the slave surface is adjusted. Displacements associated with adjusting the surface to remove overclosures do not cause any initial strain or stress for contact defined in the first step of the analysis. When conflicting constraints exist, initial overclosures may not be completely resolved by repositioning nodes. In this case severe mesh distortions can result near the beginning of an analysis when the contact pair algorithm is used. The general contact algorithm stores any unresolved initial penetrations as offsets to avoid large initial accelerations.

In subsequent steps any nodal adjustments to remove initial overclosures cause strains that often cause severe mesh distortions because the entire nodal adjustments occur in a single, very brief increment. This is especially true when the kinematic constraint method is used. For example, if a node is overclosed by 1.0×10^{-3} m and the increment time is 1.0×10^{-7} s, the acceleration applied to the node to correct the overclosure is 2.0×10^{11} m/s^2. Such a large acceleration applied to a single node typically will cause warnings about deformation speed exceeding the wave speed of the material and warnings about severe mesh distortions a few increments later, once the large acceleration has deformed the associated elements significantly. Even a very slight initial overclosure can induce extremely large accelerations for kinematic contact. In general, it is important that in the second step and beyond any new contact surfaces that you define are not overclosed.

Figure 12–47 shows a common case of initial overclosure of two contact surfaces. All of the nodes on the contact surfaces lie exactly on the same arc of a circle; but since the mesh of the inner surface is finer than that of the outer surface and since the element edges are linear, some nodes on the finer, inner surface initially penetrate the outer surface. Assuming that the pure master-slave approach is used, Figure 12–48 shows the initial, strain-free displacements applied to the slave-surface nodes by Abaqus/Explicit. In the absence of external loads this geometry is stress free. If the default, balanced master-slave approach is used, a different initial set of displacements is obtained, and the resulting mesh is not entirely stress free.

12.10 Abaqus/Explicit example: circuit board drop test

In this example you will investigate the behavior of a circuit board in protective crushable foam packaging dropped at an angle onto a rigid surface. Your goal is to assess whether the foam packaging is adequate to prevent circuit board damage when the board is dropped from a height of 1 meter. You will use the general contact capability in Abaqus/Explicit to model the interactions between the different components.

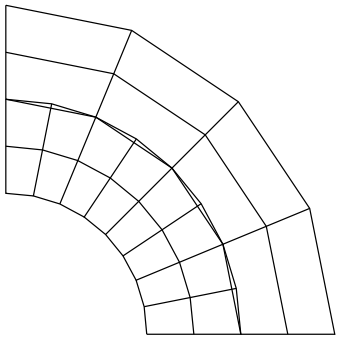


Figure 12–47 Original overclosure of two contact surfaces.

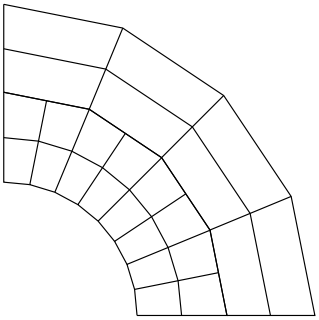


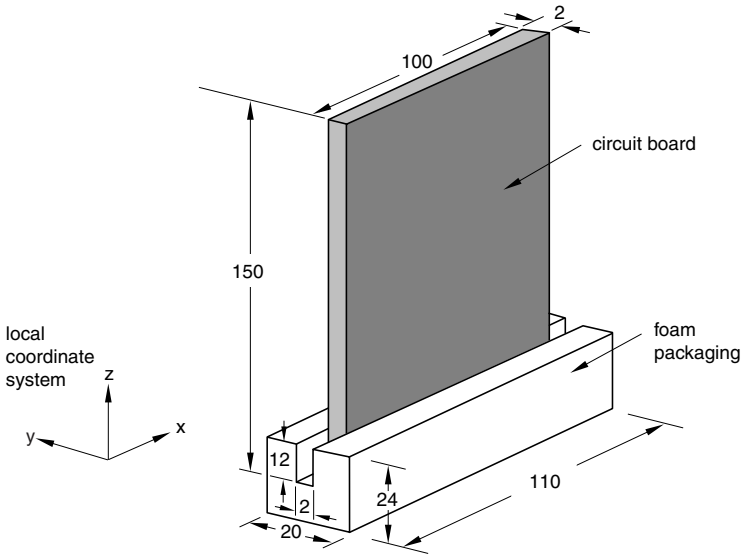
Figure 12–48 Corrected contact surfaces.

Figure 12–49 shows the dimensions of the circuit board and foam packaging in millimeters and the material properties.

12.10.1 Coordinate system

While the circuit board will be dropped at an angle, it is easiest to use the *SYSTEM option to define the mesh aligned with a local rectangular coordinate system, as shown in Figure 12–49. The *SYSTEM option transforms nodal coordinates from the local coordinate system to the global coordinate system. This option allows you to define the circuit board in the x–z plane of the local coordinate system, which is rotated by the desired angle relative to the global coordinate system.

The *SYSTEM option defines a new coordinate system by specifying three points: a local origin, a point on the local x-axis, and a point in the local x–y plane. Before defining the nodes for the circuit board, use the following option to tilt the mesh so that it lands on its corner:



Material properties

Circuit board material (plastic):

$E = 45 \times 10^9$ Pa

$\nu = 0.3$

$\rho = 500$ kg/m^3

Foam packaging material is crushable foam:

$E = 3 \times 10^6$ Pa

$\nu = 0.0$

$\rho = 100$ kg/m^3

(Foam plasticity data are given in the text.)

Figure 12–49 Dimensions in millimeters and material properties.

```
*SYSTEM
0., 0., 0., .5, .707, .25
-.5, .707, -.5
```

All subsequent nodal definitions will be in this local coordinate system. To reset the coordinate system to the default, use another *SYSTEM option with no data lines.

12.10.2 Mesh design

The overall mesh for this problem is shown in Figure 12–50.

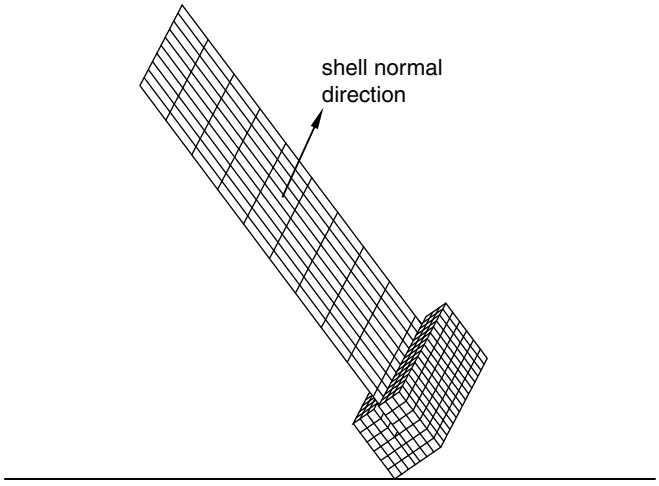


Figure 12–50 Mesh of the circuit board and foam packaging.

Define the circuit board so that the shell normals are in the direction indicated. Defining the bottom corner of the foam packaging as the origin of your model will ensure the correct positioning of the circuit board and packaging. Since the ground onto which the board will be dropped is effectively rigid, use a single R3D4 element for this part of the model. The packaging is a three-dimensional solid structure that should be modeled using C3D8R elements. The circuit board itself can be considered as a thin, flat plate with various chips attached to it. Therefore, model the circuit board with S4R elements, and model the chips with MASS elements.

Since you will be using shell elements for the circuit board, Abaqus/Explicit will, by default, use the original shell element thickness when checking for contact. The circuit board and its slot in the foam packaging are both the same thickness (2 mm) so that there is a snug fit between the two bodies. In this example the circuit board is a mesh of 10 × 10 S4R elements, and the foam packaging is a mesh of 6 × 7 × 15 elements, as shown in Figure 12–51.

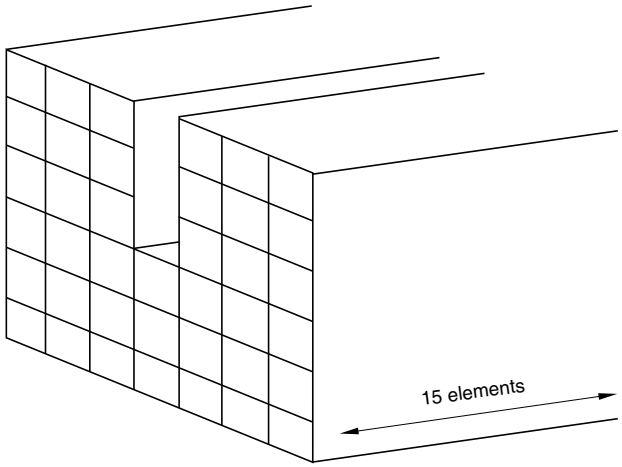


Figure 12–51 Packaging mesh detail.

The MASS elements are positioned as shown in Figure 12–52. The mesh for the packaging is too coarse near the impacting corner to provide highly accurate results. However, the mesh is adequate for a low-cost preliminary study.

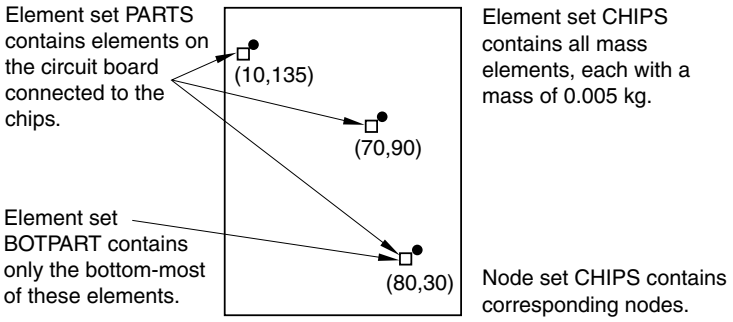

Element set PARTS contains elements on the circuit board connected to the chips.
(10,135)
(70,90)
Element set BOTPART contains only the bottom-most of these elements.
(80,30)
Element set CHIPS contains all mass elements, each with a mass of 0.005 kg.
Node set CHIPS contains corresponding nodes.

Figure 12–52 Position of mass elements on circuit board. Numbers in parentheses are (*x, y*) coordinates in millimeters based on a local origin at the bottom left-hand corner of the circuit board.

12.10.3 Node and element sets

The steps that follow assume that you have access to the full input file for this example. This input file, **circuit.inp**, is provided in "Circuit board drop test," Section A.15, in the online HTML version of this manual. Instructions on how to fetch and run the script are given in Appendix A, "Example Files."

Figure 12–52 and Figure 12–53 show all of the sets necessary to apply the element properties, loads, initial conditions, and boundary conditions, as well as to request output for postprocessing.

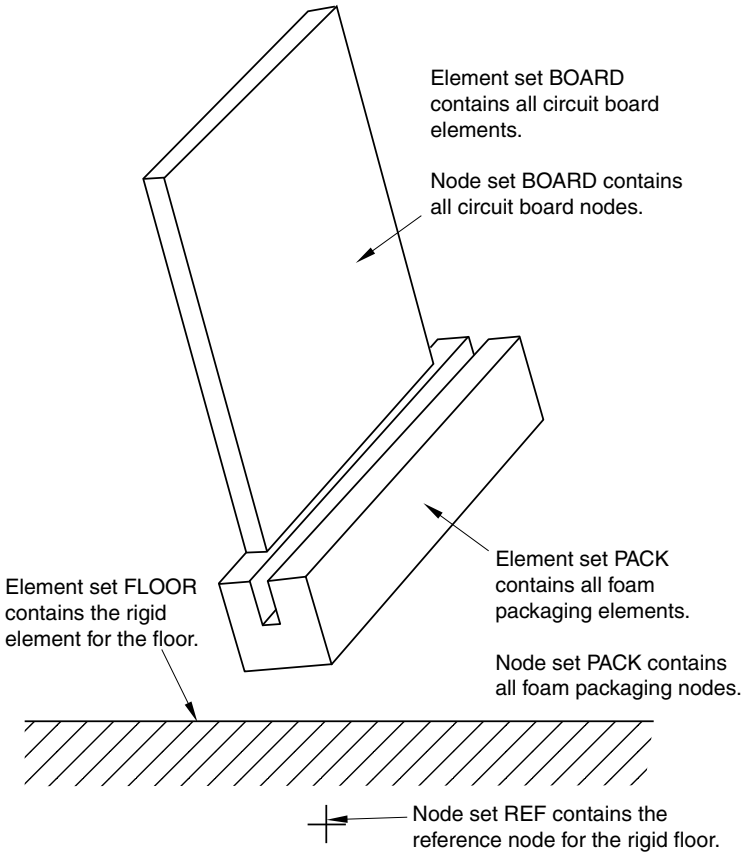

Element set BOARD contains all circuit board elements.
Node set BOARD contains all circuit board nodes.
Element set PACK contains all foam packaging elements.
Node set PACK contains all foam packaging nodes.
Element set FLOOR contains the rigid element for the floor.
Node set REF contains the reference node for the rigid floor.

Figure 12–53 Necessary node and element sets.

Include the circuit board elements in an element set called **BOARD**, and include the corresponding circuit board nodes in a node set called **BOARD**. Similarly, for the foam packaging include the elements in an

element set called **PACK**, and include the nodes in a node set called **PACK**. The element sets will be used to refer to material properties, and the node sets will be used to apply initial conditions. Create an element set called **FLOOR** containing the floor's rigid element and a node set called **REF** containing the reference node for the rigid surface modeling the floor. Include the mass elements modeling the chips in an element set called **CHIPS**.

12.10.4 Simulating free fall

Two methods could be used to simulate the circuit board being dropped from a height of 1 meter. You could model the circuit board and foam at a height of 1 meter above the rigid surface and allow Abaqus/Explicit to calculate the motion under the influence of gravity; however, this method is clearly impractical because of the large number of increments required to complete the "free-fall" part of the simulation. The most efficient method is to model the circuit board and packaging in an initial position very close to the surface with an initial velocity to simulate the 1 meter drop (4.43 m/s).

12.10.5 Reviewing the input file—the model data

We now review the model data required for this simulation, including the model description, the node and element definitions, element and material properties, boundary and initial conditions, and surface definitions. You can review these data by fetching and opening the input file **circuit.inp**.

Model description

The *HEADING option in this example provides a suitable heading for your model. SI units are used in this example.

```
*HEADING
Circuit board drop test
1.0 meter drop
SI units (kg, m, s, N)
```

Nodal coordinates and element connectivity

Use your preprocessor to generate the mesh in the local coordinate system. Precede the nodal definitions with the *SYSTEM option to transform the nodes into the tilted coordinate system, as described previously. In **circuit.inp**, the nodal definitions for the foam packaging and circuit board look like

```
*SYSTEM
0., 0., 0., .5, .707, .25
-.5, .707, -.5
*NODE
1,      0.005, -0.010, 0.012
11,     0.005, -0.010, 0.162
```

```
.
.
** Reset coordinate system
**
*SYSTEM
```

When you have finished defining the nodes in the rotated, local coordinate system, use the ∗SYSTEM option again without any data lines so that additional node numbers will be given in the global coordinate system. Define the nodes for the rigid surface so that it is large enough to keep the deformable bodies from falling off any of its edges. Use a 0.1 mm vertical clearance from the bottom corner of the foam packaging to ensure that there is no initial overclosure of the contact surfaces.

Element properties

Give each element set appropriate section properties. Include the appropriate MATERIAL parameter on each section option so that each set of elements is linked to a material definition. We have named the foam packaging material **FOAM**, and we will define it in the next section.

```
*SOLID SECTION, ELSET=PACK, MATERIAL=FOAM, CONTROLS=HGLASS
*SECTION CONTROLS, NAME=HGLASS, HOURGLASS=ENHANCED
```

For the circuit board it is most meaningful to output stress results in the longitudinal and lateral directions, aligned with the edges of the board. Therefore, we need to specify local material directions for the circuit board mesh. We can use the same local coordinate system that we previously defined using the ∗SYSTEM option. The desired material directions can be achieved using the ∗ORIENTATION option with the DEFINITION=COORDINATES parameter. On the first data line specify the *x*-, *y*-, and *z*-coordinates of two points, *a* and *b*, respectively, to define the local coordinate system. On the second data line specify an additional rotation of 90° about the local 2- (or *y*-) axis. The name of the ORIENTATION is then referred to on the ∗SHELL SECTION option.

```
*SHELL SECTION, ELSET=BOARD, MATERIAL=PCB, ORIENTATION=OR1
0.002,
*ORIENTATION, NAME=OR1, SYSTEM=RECTANGULAR,
DEFINITION=COORDINATES
0.5, 0.707, 0.25, -0.5, 0.707, -0.5
2, 90.0
```

The mass of each of the chips on the circuit board is defined to be 0.005 kg using the ∗MASS option.

```
*MASS, ELSET=CHIPS
0.005,
```

Define the rigid body by referring to the element set **FLOOR** and the rigid body reference node on the ∗RIGID BODY option. The actual node *number* of the reference node must be specified, not the node set name.

```
*RIGID BODY, ELSET=FLOOR, REF NODE=<reference node number>
```

Material properties

We You now need to define the material properties for the circuit board and the foam packaging. For the circuit board use a PCB elastic material with a Young's modulus of 45 GPa, a Poisson's ratio of 0.3, and a density of 500 kg/m^3.

```
*MATERIAL, NAME=PCB
*ELASTIC
45.E9, 0.3
*DENSITY
500.,
```

The foam packaging material is modeled using the crushable foam plasticity model. Use the ∗ELASTIC option to define the Young's modulus as 3 MPa and the Poisson's ratio as 0.0. The material density is 100 kg/m^3.

```
*MATERIAL, NAME=FOAM
*ELASTIC
 3.E6, 0.0
*DENSITY
 100.,
```

The yield surface of a crushable foam in the p–q (pressure stress–Mises equivalent stress) plane is illustrated in Figure 12–54. The ∗CRUSHABLE FOAM, HARDENING=VOLUMETRIC option uses two data items to define the initial yield behavior.

```
*CRUSHABLE FOAM, HARDENING=VOLUMETRIC
 1.1, 0.1
```

The first data item is the the ratio of initial yield stress in uniaxial compression to initial yield stress in hydrostatic compression, σ_c^0/p_c^0; we have chosen it to be 1.1. The second data item is the ratio of yield stress in hydrostatic tension to initial yield stress in hydrostatic compression, p_t/p_c^0. This data item is given as a positive value; in this problem we have chosen it to be 0.1.

Include hardening effects with the ∗CRUSHABLE FOAM HARDENING option. The first data item on each line is the yield stress in uniaxial compression, given as a positive value; the second data item on each line is the absolute value of the corresponding plastic strain. The crushable foam hardening model follows the curve shown in Figure 12–55.

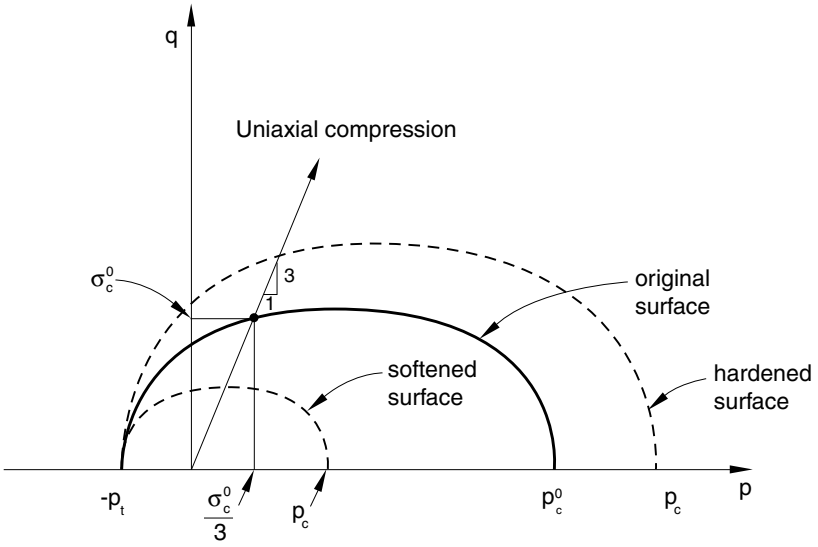


Figure 12–54 Crushable foam model: yield surface in the p–q plane.

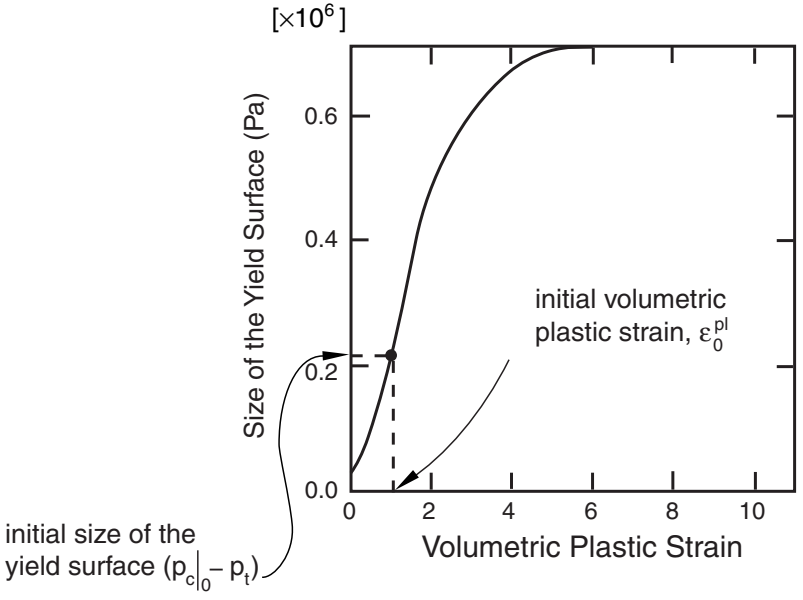


Figure 12–55 Foam hardening material data.

```
*CRUSHABLE FOAM HARDENING
0.22000E6, 0.0
0.24651E6, 0.1
0.27294E6, 0.2
0.29902E6, 0.3
0.32455E6, 0.4
0.34935E6, 0.5
0.37326E6, 0.6
0.39617E6, 0.7
0.41801E6, 0.8
0.43872E6, 0.9
0.45827E6, 1.0
0.49384E6, 1.2
0.52484E6, 1.4
0.55153E6, 1.6
0.57431E6, 1.8
0.59359E6, 2.0
0.62936E6, 2.5
0.65199E6, 3.0
0.68334E6, 5.0
0.68833E6, 10.0
```

Boundary conditions

The rigid surface representing the floor is fully constrained by applying a fixed boundary condition to the reference node, which was previously defined as node set **REF**.

```
*BOUNDARY
REF, ENCASTRE
```

Initial conditions

The circuit board and foam packaging is given an initial velocity of −4.43 m/s in the global 3-direction, corresponding to the velocity at the end of a 1 meter free fall.

```
*INITIAL CONDITIONS, TYPE=VELOCITY
BOARD, 3, -4.43
PACK, 3, -4.43
```

Defining contact

Either contact algorithm could be used for this problem. However, the definition of contact using the contact pair algorithm would be more cumbersome since, unlike general contact, the surfaces involved in contact pairs cannot span more than one body. We use the general contact algorithm in this example to demonstrate the simplicity of the contact definition for more complex geometries.

First, a named contact property is defined using the *SURFACE INTERACTION option, a friction coefficient of 0.3 is defined.

```
*SURFACE INTERACTION, NAME=FRIC
*FRICTION
0.3,
```

Use the *CONTACT option to define a general contact interaction. Use the ALL EXTERIOR parameter on the *CONTACT INCLUSIONS option to specify self-contact for the unnamed, all-inclusive surface defined automatically by Abaqus/Explicit. The *CONTACT PROPERTY ASSIGNMENT option is used to assign the contact property named **FRIC** to the general contact interaction.

```
*CONTACT
*CONTACT INCLUSIONS, ALL EXTERIOR
*CONTACT PROPERTY ASSIGNMENT
 , , FRIC
```

12.10.6 Reviewing the input file—the history data

The *DYNAMIC, EXPLICIT option is used to select a dynamic stress/displacement analysis using explicit integration. The time period of the step is defined as 20 ms.

```
*STEP
*DYNAMIC, EXPLICIT
, 0.02
```

Output requests

The preselected field data are written to the output database file by including the following line in the input file:

```
*OUTPUT, FIELD, VARIABLE=PRESELECT
```

Values of vertical nodal displacement (U3), velocity (V3), and acceleration (A3) will be written for each of the attached chips as history data to the output database file. An output interval of 0.07 ms has been selected.

```
*OUTPUT, HISTORY, TIME INTERVAL=0.07E-3
*NODE OUTPUT, NSET=CHIPS
U3, V3, A3
```

Energy values will be written summed over the entire model. Specifically, write values for kinetic energy (ALLKE), internal energy (ALLIE), elastic strain energy (ALLSE), artificial energy (ALLAE), and the energy dissipated by plastic deformation (ALLPD).

```
*ENERGY OUTPUT
ALLIE, ALLKE, ALLPD, ALLAE, ALLSE
```

The end of the step is indicated with the *END STEP option.

12.10.7 Running the analysis

Run the analysis using the following command:

```
abaqus job=circuit analysis
```

This analysis is somewhat more complicated than the previous analyses in this guide, and it may take 45 minutes or more to run to completion, depending on the power of your computer.

Status file

Information concerning the initial stable time increment can be found at the top of the status file. The 10 most critical elements (i.e., those resulting in the smallest time increments) are also shown in rank order.

```
 ---------------------------------------------------------------------------------
 MODEL INFORMATION (IN GLOBAL X-Y COORDINATES)
---------------------------------------------------------------------------------

   Total mass in model = 3.49594E-02
   Center of mass of model = (-1.076765E-02, 4.948691E-02, 8.492255E-02)

    Moments of Inertia :
                About Center of Mass              About Origin
      I(XX)          6.655668E-05                 4.042925E-04
      I(YY)          9.949297E-05                 3.556680E-04
      I(ZZ)          6.893156E-05                 1.585989E-04
      I(XY)         -1.344118E-05                 5.187227E-06
      I(YZ)         -5.240504E-06                -1.521594E-04
      I(ZX)          3.958677E-05                 7.155426E-05

---------------------------------------------------------------------------------
 STABLE TIME INCREMENT INFORMATION
---------------------------------------------------------------------------------

  The stable time increment estimate for each element is based on
  linearization about the initial state.

   Initial time increment = 8.80392E-07

   Statistics for all elements:
      Mean = 1.04795E-05
      Standard deviation = 3.99235E-06

   Most critical elements :
    Element number   Rank    Time increment   Increment ratio
   ------------------------------------------------------------
          98           1       8.803920E-07      1.000000E+00
          83           2       8.803923E-07      9.999997E-01
          80           3       8.803923E-07      9.999996E-01
          79           4       8.803925E-07      9.999995E-01
          71           5       8.803925E-07      9.999994E-01
          30           6       8.803926E-07      9.999993E-01
          36           7       8.803926E-07      9.999993E-01
          69           8       8.803926E-07      9.999993E-01
          77           9       8.803926E-07      9.999993E-01
          86          10       8.803926E-07      9.999993E-01
```

```
                                    :
                                    :
                                    :
------------------------------------------------------------------------------
 SOLUTION PROGRESS
------------------------------------------------------------------------------

 STEP 1  ORIGIN 0.0000

  Total memory used for step 1 is approximately 3.7 megabytes.
  Global time estimation algorithm will be used.
  Scaling factor:  1.0000
  Variable mass scaling factor at zero increment:  1.0000
               STEP      TOTAL         CPU       STABLE   CRITICAL      KINETIC
INCREMENT      TIME       TIME        TIME    INCREMENT    ELEMENT       ENERGY
        0  0.000E+00 0.000E+00   00:00:00 8.394E-07          98   3.430E-01
Results number 0 at increment zero.
ODB Field Frame Number       0 of      5 requested intervals at increment zero.
     1188  1.000E-03 1.000E-03   00:00:03 8.394E-07          91   3.123E-01
                                    :
                                    :
                                    :
```

12.10.8 Postprocessing

Start Abaqus/Viewer by typing the following:

```
abaqus viewer odb=circuit
```

at the operating system prompt.

Checking material directions

The material directions obtained from this orientation definition can be checked with Abaqus/Viewer.

To plot the material orientation:

1. First, change the view to a more convenient setting. If it is not visible, display the **Views** toolbar by selecting **View→Toolbars→Views** from the main menu bar. In the **Views** toolbar, select the X–Z setting.
2. From the main menu bar, select **Plot→Material Orientations→On Deformed Shape**.

   The orientations of the material directions for the circuit board at the end of the simulation are shown. The material directions are drawn in different colors. The material 1-direction is blue, the material 2-direction is yellow, and the 3-direction, if it is present, is red.
3. To view the initial material orientation, select **Result→Step/Frame**. In the **Step/Frame** dialog box that appears, select **`Increment 0`**. Click **Apply**.

   Abaqus displays the initial material directions.
4. To restore the display to the results at the end of the analysis, select the last increment available in the **Step/Frame** dialog box; and click **OK**.

Animation of results

You will create a time-history animation of the deformation to help you visualize the motion and deformation of the circuit board and foam packaging during impact.

To create a time-history animation:

1. Plot the deformed model shape at the end of the analysis.
2. From the main menu bar, select **Animate→Time History**.
   The animation of the deformed model shape begins.
3. From the main menu bar, select **View→Parallel** to turn off perspective.
4. In the context bar, click to pause the animation after a full cycle has been completed.
5. In the context bar, click and then select a node on the foam packaging near one of the corners that impacts the floor. When you restart the animation the camera will move with the selected node. If you zoom in on the node, it will remain in view throughout the animation.

   Note: To reset the camera to follow the global coordinate system, click in the context bar.

While you view the deformation history of the drop test, take note of when the foam is in contact with the floor. You should observe that the initial impact occurs over the first 4 ms of the analysis. A second impact occurs from about 8 ms to 15 ms. The deformed state of the foam and board at approximately 4 ms after impact is shown in Figure 12–56.

Plotting model energy histories

Plot graphs of various energy variables versus time. Energy output can help you evaluate whether an Abaqus/Explicit simulation is predicting an appropriate response.

To plot energy histories:

1. In the Results Tree, click mouse button 3 on **History Output** for the output database named **`circuit.odb`**. From the menu that appears, select **Filter**.
2. In the filter field, enter **`*ALL*`** to restrict the history output to just the energy output variables.
3. Select the ALLAE output variable, and save the data as **`Artificial Energy`**.
4. Select the ALLIE output variable, and save the data as **`Internal Energy`**.
5. Select the ALLKE output variable, and save the data as **`Kinetic Energy`**.
6. Select the ALLPD output variable, and save the data as **`Plastic Dissipation`**.
7. Select the ALLSE output variable, and save the data as **`Strain Energy`**.

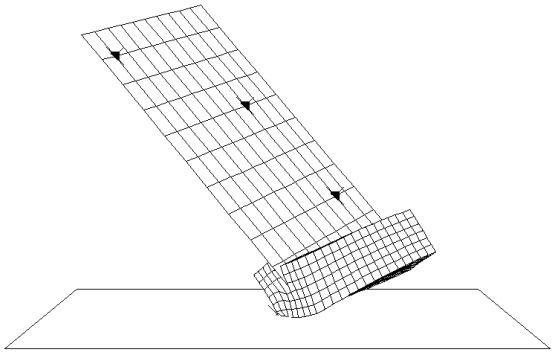


Figure 12–56 Deformed mesh at 4 ms.

8. In the Results Tree, expand the **XYData** container.
9. Select all five curves. Click mouse button 3, and select **Plot** from the menu that appears to view the *X*–*Y* plot.

   Next, you will customize the appearance of the plot; begin by changing the line styles of the curves.
10. Open the **Curve Options** dialog box.
11. In this dialog box, apply different line styles and thicknesses to each of the curves displayed in the viewport.

    Next, reposition the legend so that it appears inside the plot.
12. Double-click the legend to open the **Chart Legend Options** dialog box.
13. In this dialog box, switch to the **Area** tabbed page, and toggle on **Inset**.
14. In the viewport, drag the legend over the plot.

    Now change the format of the *X*-axis labels.
15. In the viewport, double-click the *X*-axis to access the **X Axis** options in the **Axis Options** dialog box.
16. In this dialog box, switch to the **Axes** tabbed page, and select the **Engineering** label format for the *X*-axis.

    The energy histories appear as shown in Figure 12–57.

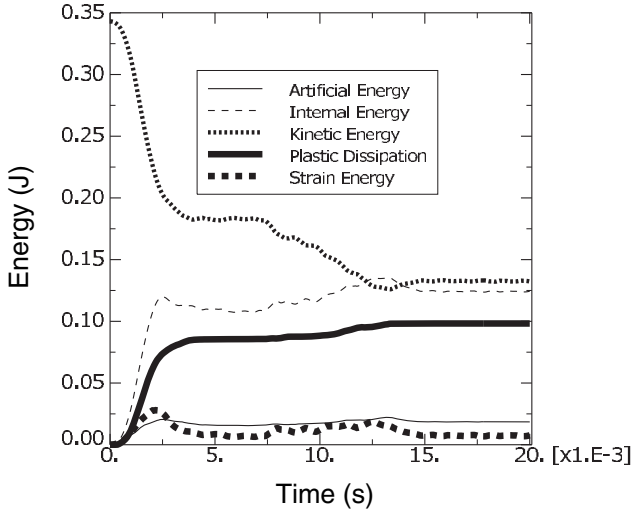


Figure 12–57 Energy results versus time.

First, consider the kinetic energy history. At the beginning of the simulation the components are in free fall, and the kinetic energy is large. The initial impact deforms the foam packaging, thus reducing the kinetic energy. The components then bounce and rotate about the impacted corner until the opposite side of the foam packaging impacts the floor at about 8 ms, further reducing the kinetic energy.

The deformation of the foam packaging during impact causes a transfer of kinetic energy to internal energy in the foam packaging and the circuit board. From Figure 12–57 we can see that the internal energy increases as the kinetic energy decreases. In fact, the internal energy is composed of elastic energy and plastically dissipated energy, both of which are also plotted in Figure 12–57. Elastic energy rises to a peak and then falls as the elastic deformation recovers, but the plastically dissipated energy continues to rise as the foam is deformed permanently.

Another important energy output variable is the artificial energy, which is a substantial fraction (approximately 15%) of the internal energy in this analysis. By now you should know that the quality of the solution would improve if the artificial energy could be decreased to a smaller fraction of the total internal energy.

What causes high artificial strain energy in this problem?

Contact at a single node—such as the corner impact in this example—can cause hourglassing, especially in a coarse mesh. Two common strategies for reducing the artificial strain energy are to refine the mesh or to round the impacting corner. For the current exercise, however, we shall continue with the original mesh, realizing that improving the mesh would lead to an improved solution.

Evaluating acceleration histories at the chips

The next result we wish to examine is the acceleration of the chips attached to the circuit board. Excessive accelerations during impact may damage the chips. Therefore, in order to assess the desirability of the foam packaging, we need to plot the acceleration histories of the three chips. Since we expect the accelerations to be greatest in the 3-direction, we will plot the variable A3.

To plot acceleration histories:

1. In the Results Tree, filter the **History Output** container according to **`*A3*`**, select the acceleration **`A3`** of the nodes **`60`**, **`357`**, and **`403`** in the set **`CHIPS`**; and plot the three *X*–*Y* data objects.

   The *X*–*Y* plot appears in the viewport. As before, customize the plot appearance to obtain a plot similar to Figure 12–58.

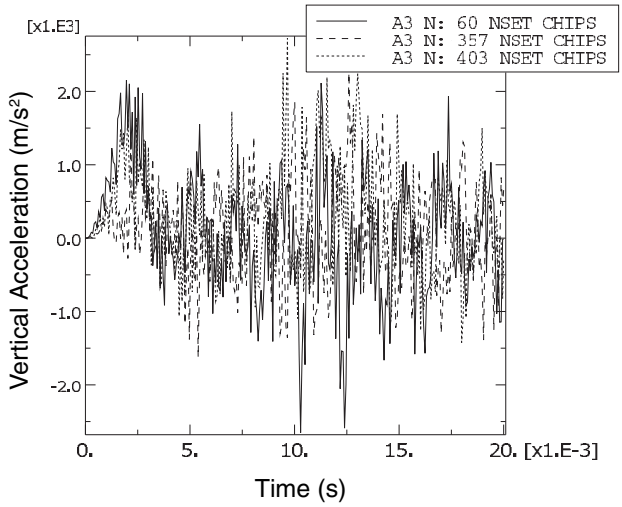


Figure 12–58 Acceleration of the three chips in the *Z*-direction.

Next, we will evaluate the plausibility of the acceleration data recorded at the bottom chip. To do this, we will integrate the acceleration data to calculate the chip velocity and displacement and compare the results to the velocity and displacement data recorded directly by Abaqus/Explicit.

To integrate the bottom chip acceleration history:

1. In the Results Tree, filter the **History Output** container according to **`*Node 403*`**, select the acceleration **`A3`** of node **`403`**; and save the data as **`A3`**.

2. In the Results Tree, double-click **XYData**; then select **Operate on XY data** in the **Create XY Data** dialog box. Click **Continue**.
3. In the **Operate on XY Data** dialog box, integrate acceleration **A3** to calculate velocity and subtract the initial velocity magnitude of 4.43 m/s. The expression at the top of the dialog box should appear as:

```
integrate ( "A3" ) - 4.43
```

4. Click **Plot Expression** to plot the calculated velocity curve.
5. In the Results Tree, click mouse button 3 on the velocity **V3** history output for node **403**; and select **Add to Plot** from the menu that appears.

   The *X*–*Y* plot appears in the viewport. As before, customize the plot appearance to obtain a plot similar to Figure 12–59. The velocity curve you produced by integrating the acceleration data may be different from the one pictured here. The reason for this will be discussed later.

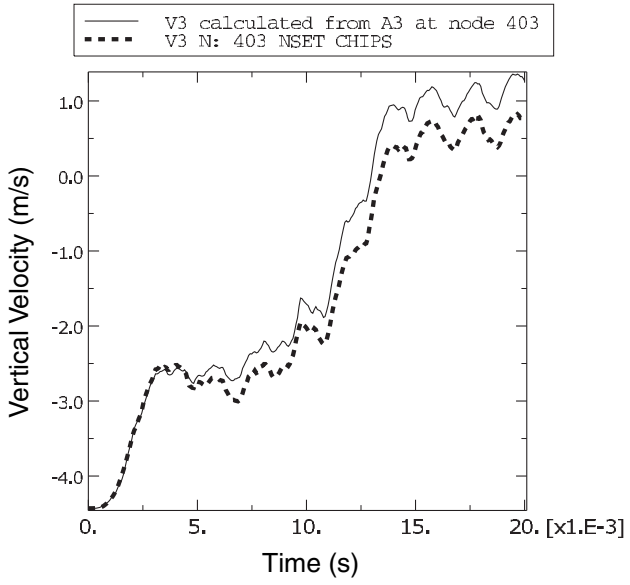


Figure 12–59 Velocity the bottom chip in the *Z*-direction.

6. In the **Operate on XY Data** dialog box, integrate acceleration **A3** a second time to calculate chip displacement. The expression at the top of the dialog box should appear as:

```
integrate ( integrate ( "A3" ) - 4.43 )
```

7. Click **Plot Expression** to plot the calculated displacement curve.

Notice that the *Y*-value type is length. In order to plot the calculated displacement with the same *Y*-axis as the displacement output recorded during the analysis, we must save the *X*–*Y* data and change the *Y*-value type to displacement.

8. Click **Save As** to save the calculated displacement curve as **U3-from-A3**.
9. In the **XYData** container of the Results Tree, click mouse button 3 on **U3-from-A3**; and select **Edit** from the menu that appears.
10. In the **Edit XY Data** dialog box, choose **Displacement** as the *Y*-value type.
11. In the Results Tree, double-click **U3-from-A3** to recreate the calculated displacement plot with the displacement *Y*-value type.
12. In the Results Tree, click mouse button 3 on the displacement **U3** history output for node **403**; and select **Add to Plot** from the menu that appears.

    The *X*–*Y* plot appears in the viewport. As before, customize the plot appearance to obtain a plot similar to Figure 12–60. Again, the curve you produced by integrating the acceleration data may be different from the one pictured here. The reason for this will be discussed later.

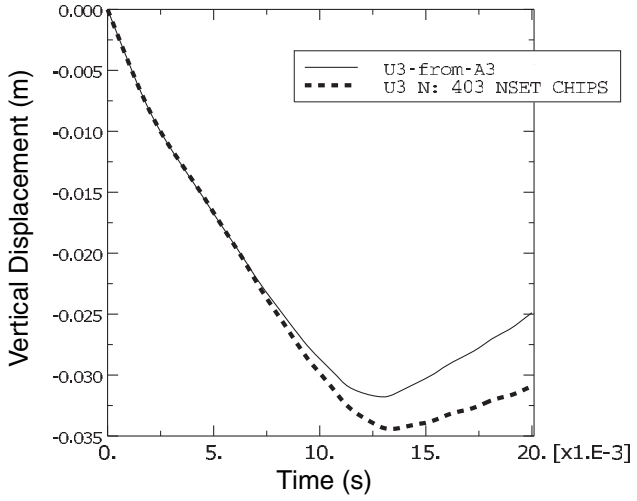


Figure 12–60 Displacement of the bottom chip in the *Z*-direction.

Why are the velocity and displacement curves calculated by integrating the acceleration data different from the velocity and displacement recorded during the analysis?

In this example the acceleration data has been corrupted by a phenomenon called aliasing. Aliasing is a form of data corruption that occurs when a signal (such as the results of an Abaqus analysis) is sampled at a series of discrete points in time, but not enough data points are saved in order to correctly describe the signal. The aliasing phenomenon can be addressed using digital signal

processing (DSP) methods, a fundamental principle of which is the Nyquist Sampling Theorem (also known as the Shannon Sampling Theorem). The Sampling Theorem requires that a signal be sampled at a rate that is greater than twice the signal's highest frequency. Therefore, the maximum frequency content that can be described by a given sampling rate is half that rate (the Nyquist frequency). Sampling (storing) a signal with large-amplitude oscillations at frequencies greater than the Nyquist frequency of the sample rate may produce significantly distorted results due to aliasing. In this example the chip acceleration was sampled every 0.07 ms, which is a sampling rate of 14.3 kHz (the sample rate is the inverse of the sample size). The recorded data was aliased because the chip acceleration response has frequency content above 7.2 kHz (half the sample rate).

Aliasing of a sine wave

To better understand how aliasing distorts data, consider a 1 kHz sine wave sampled using various sampling rates, as shown in Figure 12–61.

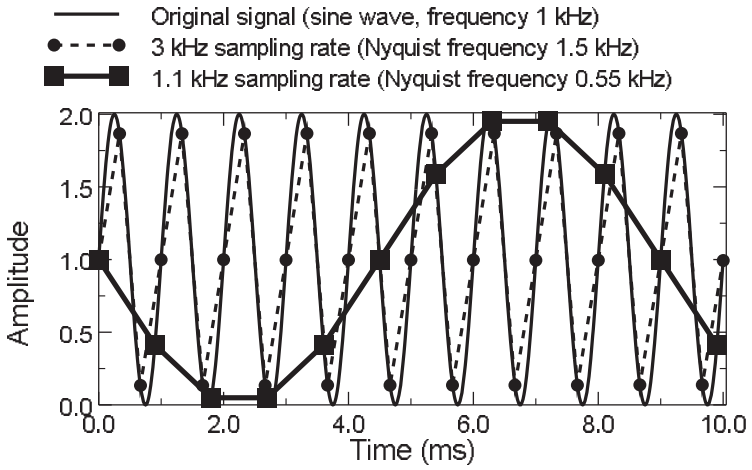


Figure 12–61 1 kHz sine wave sampled at 1.1 kHz and 3 kHz.

According to the Sampling Theorem, this signal must be sampled at a rate greater than 2 kHz to avoid alias distortions. We will evaluate what happens when the sample rate is greater than or less than this value.

Consider the data recorded with a sample rate of 1.1 kHz; this rate is less than the required 2 kHz rate. The resulting curve exhibits alias distortions because it is an extremely misleading representation of the original 1 kHz sine wave.

Now consider the data recorded with a sample rate of 3 kHz; this rate is greater than the required 2 kHz rate. The frequency content of the original signal is captured without aliasing. However, this sample rate is not high enough to guarantee that the peak values of the sampled signal are captured

very accurately. To guarantee 95% accuracy of the recorded local peak values, the sampling rate must exceed the signal frequency by a factor of ten or more.

Avoiding aliasing

In the previous two examples of aliasing (the aliased chip acceleration and the aliased sine wave), it would not have been obvious from the aliased data alone that aliasing had occurred. In addition, there is no way to uniquely reconstruct the original signal from the aliased data alone. Therefore, care should be taken to avoid aliasing your analysis results, particularly in situations when aliasing is most likely to occur.

Susceptibility to aliasing depends on a number of factors, including output rate, output variable, and model characteristics. Recall that signals with large-amplitude oscillations at frequencies greater than half the sampling rate (the Nyquist frequency) may be significantly distorted due to aliasing. The two output variables that are most likely to have large-amplitude high-frequency content are accelerations and reaction forces. Therefore, these variables are the most susceptible to aliasing. Displacements, on the other hand, are lower in frequency content by nature, so they are much less susceptible to aliasing. Other result variables, such as stress and strain, fall somewhere in between these two extremes. Any model characteristic that reduces the high-frequency response of the solution will decrease the analysis's susceptibility to aliasing. For example, an elastically dominated impact problem would be even more susceptible to aliasing than this circuit board drop test which includes energy absorbing packaging.

The safest way to ensure that aliasing is not a problem in your results is to request output at every increment. When you do this, the output rate is determined by the stable time increment, which is based on the highest possible frequency response of the model. However, requesting output at every increment is often not practical because it would result in very large output files. In addition, output at every increment is usually much more data than you need; there is no need to capture high-frequency solution noise when what you are really interested in is the lower-frequency structural response. An alternative method for avoiding aliasing is to request output at a lower rate and use the Abaqus/Explicit real-time filtering capabilities to remove high-frequency content from the result before writing data to the output database file. This technique uses less disk space than requesting output every increment; however, it is up to you to ensure that your output rate and filter choices are appropriate (to avoid aliasing or other distortions related to digital signal processing).

Abaqus/Explicit offers filtering capabilities for both field and history data. Filtering of history data only is discussed here.

12.10.9 Rerunning the analysis with output filtering

In this section you will add real-time filters to the history output requests for the circuit board drop test analysis. While Abaqus/Explicit does allow you to create user-defined output filters (Butterworth, Chebyshev Type I, and Chebyshev Type II) based on criteria that you specify, in this example we will use the built-in anti-aliasing filter. The built-in anti-aliasing filter is designed to give you the best un-aliased representation of the results recorded at the output rate you specify on the output request.

To do this, Abaqus/Explicit internally applies a low-pass, second-order, Butterworth filter with a cutoff frequency set to one-sixth of the sampling rate. For more information, see "Overview of filtering Abaqus history output" in the Dassault Systèmes Knowledge Base at www.3ds.com/support/knowledge-base or the SIMULIA Online Support System, which is accessible through the **My Support** page at www.simulia.com. For more information on defining your own real-time filters, see "Filtering output and operating on output in Abaqus/Explicit" in "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual.

Modifying the history output requests

When Abaqus writes nodal history output to the output database, it gives each data object a name that indicates the recorded output variable, the filter used (if any), the node number, and the node set. For this exercise you will be creating multiple output requests for the bottom-most chip (node **403**) that differ only by the output sample rate, which is not a component of the history output name. To easily distinguish between the similar output requests, create two new sets for node **403**. Name one of the new sets **BotChip-all** and the other **BotChip-largeInc**.

```
*NSET, NSET=BotChip-all
403
*NSET, NSET=BotChip-largeInc
403
```

Next, add a new history output request for the vertical displacement, velocity, and acceleration of the chips. In addition, request element logarithmic strain components (LE11, LE22 and LE12), and logarithmic principal strain (LEP) at the top face (section point 5) of element set **BOTPART** of the circuit board to which the bottom-most chip is attached. For these output requests record the data at every 0.07 ms and apply the built-in anti-aliasing filter.

```
*OUTPUT, HISTORY, TIME INTERVAL=0.07E-3, FILTER=ANTIALIASING
*NODE OUTPUT, NSET=CHIPS
 U3, V3, A3
*ELEMENT OUTPUT, ELSET=BOTPART
 5,
 LE11, LE22, LE12, LEP
```

Request history output at every increment for the vertical displacement, velocity, and acceleration of the bottom-most chip. Use node set **BotChip-all** for this output request.

```
*OUTPUT, HISTORY, FREQUENCY=1
*NODE OUTPUT, NSET=BotChip-all
 U3, V3, A3
```

Add one more output request for the vertical displacement, velocity, and acceleration of the bottom-most chip. This time request the output every 0.7 ms and apply the built-in anti-aliasing filter. Use node set **BotChip-largeInc**.

```
*OUTPUT, HISTORY, TIME INTERVAL=0.7E-3, FILTER=ANTIALIASING
*NODE OUTPUT, NSET=BotChip-largeInc
 U3, V3, A3
```

When you are finished, there will be four history output requests for the bottom chip (the original one and the three added here).

Evaluating the filtered acceleration of the bottom chip

When the analysis completes, test the plausibility of the acceleration history output for the bottom chip recorded every 0.07 ms using the built-in, anti-aliasing filter. Do this by saving and then integrating the filtered acceleration data (**A3_ANTIALIASING** for node **403** in set **CHIPS**) and comparing the results to recorded velocity and displacement data, just as you did earlier for the unfiltered version of these results. This time you should find that the velocity and displacement curves calculated by integrating the filtered acceleration are very similar to the velocity and displacement values written to the output database during the analysis. You may also have noticed that the velocity and displacement results are the same regardless of whether or not the built-in anti-aliasing filter is used. This is because the highest frequency content of the nodal velocity and displacement curves is much less than half the sampling rate. Consequently, no aliasing occurred when the data was recorded without filtering, and when the built-in anti-aliasing filter was applied it had no effect because there was no high frequency response to remove.

Next, compare the acceleration **A3** history output recorded every increment with the two acceleration **A3** history curves recorded every 0.07 ms. Plot the data recorded at every increment first so that it does not obscure the other results.

To plot the acceleration histories

1. In the Results Tree, filter the **History Output** container according to ***A3*Node 403*** and double-click the acceleration **A3** history output for the node set **BotChip-all**.
2. Select the two acceleration **A3** history output objects for **Node 403** in the set **CHIPS** (one filtered with the built-in anti-aliasing filter and the other with no filtering) using [Ctrl]+Click; click mouse button 3 and select **Add to Plot** from the menu that appears.

   The *X–Y* plot appears in the viewport. Zoom in to view only the first third of the results and customize the plot appearance to obtain a plot similar to Figure 12–62.

First consider the acceleration history recorded every increment. This curve contains a lot of data, including high-frequency solution noise which becomes so large in magnitude that it obscures the structurally-significant lower-frequency components of the acceleration. When output is requested every increment, the output time increment is the same as the stable time increment, which (in order to ensure stability) is based on a conservative estimate of the highest possible frequency response of the model. Frequencies of structural significance are typically two to four orders of magnitude less than the highest frequency of the model. In this example the stable time increment ranges between 8.4×10^{-4} ms to 8.8×10^{-4} ms (see the status file, **circuit.sta**),

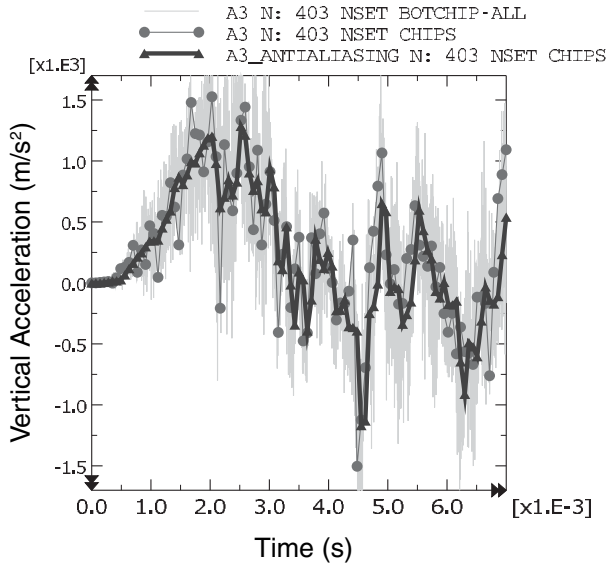


Figure 12–62 Comparison of acceleration output with and without filtering.

which corresponds to a sample rate of about 1 MHz; this sample rate has been rounded down for this discussion, even though it means that the value is not conservative. Recalling the Sampling Theorem, the highest frequency that can be described by a given sample rate is half that rate; therefore, the highest frequency of this model is about 500 kHz and typical structural frequencies could be as high as 2–3 kHz (more than 2 orders of magnitude less than the highest model frequency). While the output recorded every increment contains a lot of undesirable solution noise in the 3 to 500 kHz range, it is guaranteed to be good (not aliased) data, which can be filtered later with a postprocessing operation if necessary.

Next consider the data recorded every 0.07 ms without any filtering. Recall that this is the curve we know to be corrupted by aliasing. The curve jumps from point to point by directly including whatever the raw acceleration value happens to be after each 0.07 ms interval. The variable nature of the high-frequency noise makes this aliased result very sensitive to otherwise imperceptible variations in the solution (due to differences between computer platforms, for example), hence the results you recorded every 0.07 increments may be significantly different from those shown in Figure 12–62. Similarly, the velocity and displacement curves we produced by integrating the aliased acceleration (Figure 12–59 and Figure 12–60) data are extremely sensitive to small differences in the solution noise.

When the built-in anti-aliasing filter is applied to the output requested every 0.07 ms, frequency content that is too high to be captured by the 14.3 kHz sample rate is filtered out before the result is written to the output database. To do this, Abaqus internally defines a low-pass,

second-order, Butterworth filter. Low-pass filters attenuate the frequency content of a signal that is above a specified cutoff frequency. An ideal low-pass filter would completely eliminate all frequencies above the cutoff frequency while having no effect on the frequency content below the cutoff frequency. In reality there is a transition band of frequencies surrounding the cutoff frequency that are partially attenuated. To compensate for this, the built-in anti-aliasing filter has a cutoff frequency that is one-sixth of the sample rate, a value lower than the Nyquist frequency of one-half the sample rate. In most cases (including this example), this cutoff frequency is adequate to ensure that all frequency content above the Nyquist frequency has been removed before the data are written to the output database.

Abaqus/Explicit does not check to ensure that the specified output time interval provides an appropriate cutoff frequency for the internal anti-aliasing filter; for example, Abaqus does not check that only the noise of the signal is eliminated. When the acceleration data are recorded every 0.07 ms, the internal anti-aliasing filter is applied with a cutoff frequency of 2.4 kHz. This cutoff frequency is nearly the same value we previously determined to be the maximum physically meaningful frequency for the model (more than two orders of magnitude less than the maximum frequency the stable time increment can capture). The 0.07 ms output interval was intentionally chosen for this example to avoid filtering frequency content that could be physically meaningful. Next, we will study the results when the anti-aliasing filter is applied with a sample interval that is too large.

To plot the filtered acceleration histories

1. In the Results Tree, double-click the acceleration **A3** history output for the node set **BotChip-all**.
2. Select the two filtered acceleration **A3_ANTIALIASING** history output objects for **Node 403**; click mouse button 3 and select **Add to Plot** from the menu that appears.

   The *X–Y* plot appears in the viewport. Zoom out and customize the plot appearance to obtain a plot similar to Figure 12–63.

Figure 12–63 clearly illustrates some of the problems that can arise when the built-in anti-aliasing filter is used with too large an output time increment. First, notice that many of the oscillations in the acceleration output are filtered out when the acceleration is recorded with large time increments. In this dynamic impact problem it is likely that a significant portion of the removed frequency content is physically meaningful. Previously, we estimated that the frequency of the structural response may be as large as 2–3 kHz; however, when the sample interval is 0.7 ms, filtering is performed with a low cutoff frequency of 0.24 kHz (sample interval of 0.7 ms corresponds to a sample frequency of 1.43 kHz, one-sixth of which is the 0.24 kHz cutoff frequency). Even though the results recorded every 0.7 ms may not capture all physically meaningful frequency content, it does capture the low-frequency content of the acceleration data without distortions due to aliasing. Keep in mind that filtering decreases the peak value estimations, which is desirable if only solution noise is filtered, but can be misleading when physically meaningful solution variations have been removed.

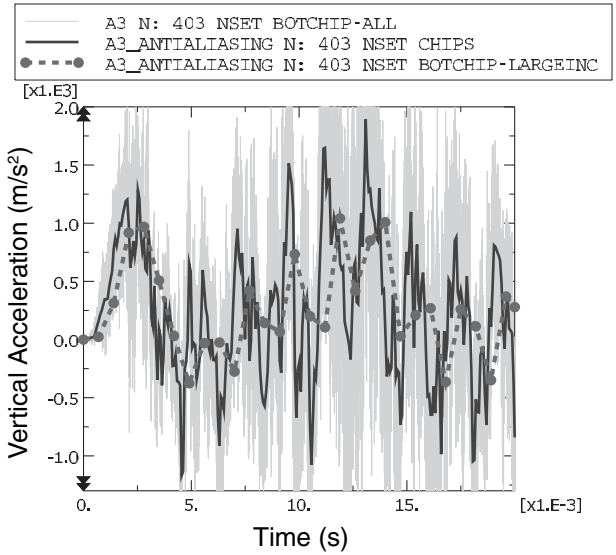


Figure 12–63 Filtered acceleration with different output sampling rates.

Another issue to note is that there is a time delay in the acceleration results recorded every 0.7 ms. This time delay (or phase shift) affects all real-time filters. The filter must have some input in order to produce output; consequently the filtered result will include some time delay. While some time delay is introduced for all real-time filtering, the time delay becomes more pronounced as the filter cutoff frequency decreases; the filter must have input over a longer span of time in order to remove lower frequency content. Increasing the filter order (an option if you have created a user-defined filter, rather than using the second-order built-in anti-aliasing filter) also results in an increase in the output time delay. For more information, see "Filtering output and operating on output in Abaqus/Explicit" in "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual.

Use the real-time filtering functionality with caution. In this example we would not have been able to identify the problems with the heavily filtered data if we did not have appropriate data for comparison. In general it is best to use a minimal amount of filtering in Abaqus/Explicit, so that the output database contains a rich, un-aliased, representation for the solution recorded at a reasonable number of time points (rather than at every increment). If additional filtering is necessary, it can be done as a postprocessing operation in Abaqus/Viewer.

Filtering acceleration history in Abaqus/Viewer

In this section we will use Abaqus/Viewer to filter the acceleration history data written to the output database. Filtering as a postprocessing operation in Abaqus/Viewer has several advantages over the real-time filtering available in Abaqus/Explicit. In the Abaqus/Viewer you can quickly filter

X–Y data and plot the results. You can easily compare the filtered results to the unfiltered results to verify that the filter produced the desired effect. Using this technique you can quickly iterate to find appropriate filter parameters. In addition, the Abaqus/Viewer filters do not suffer from the time delay that is unavoidable when filtering is applied during the analysis. Keep in mind, however, that postprocessing filters cannot compensate for poor analysis history output; if the data has been aliased or if physically meaningful frequencies have been removed, no postprocessing operation can recover the lost content.

To demonstrate the differences between filtering in Abaqus/Viewer and filtering in Abaqus/Explicit, we will filter the acceleration of the bottom chip in Abaqus/Viewer and compare the results to the filtered data Abaqus/Explicit wrote to the output database.

To filter acceleration history:

1. In the Results Tree, select the acceleration **A3** history output for the node set **BotChip-all**, and save the data as **A3-all**.
2. In the Results Tree, double-click **XYData**; then select **Operate on XY data** in the **Create XY Data** dialog box. Click **Continue**.
3. In the **Operate on XY Data** dialog box, filter **A3-all** with filter options that are equivalent to those applied by the Abaqus/Explicit built-in anti-aliasing filter when the output increment is 0.7 ms. Recall that the built-in anti-aliasing filter is a second-order Butterworth filter with a cutoff frequency that is one-sixth of the output sample rate; hence, the expression at the top of the dialog box should appear as

   ```
   butterworthFilter ( xyData="A3-all",
    cutoffFrequency=1/(6*0.0007) )
   ```
4. Click **Plot Expression** to plot the filtered acceleration curve.
5. In the Results Tree, click mouse button 3 on the filtered acceleration **A3_ANTIALIASING** history output for node set **BotChip-largeInc**; and select **Add to Plot** from the menu that appears. If you wish, also add the filtered acceleration history for node **403** in the set **CHIPS**.

   The *X–Y* plot appears in the viewport. As before, customize the plot appearance to obtain a plot similar to Figure 12–64.

In Figure 12–64 it is clear that the postprocessing filter in Abaqus/Viewer does not suffer from the time delay that occurs when filtering is performed while the analysis is running. This is because the Abaqus/Viewer filters are bidirectional, which means that the filtering is applied first in a forward pass (which introduces some time delay) and then in a backward pass (which removes the time delay). As a consequence of the bidirectional filtering in Abaqus/Viewer, the filtering is essentially applied twice, which results in additional attenuation of the filtered signal compared to the attenuation achieved with a single-pass filter. This is why the local peaks in the acceleration curve filtered in Abaqus/Viewer are a bit lower than those in the curve filtered by Abaqus/Explicit.

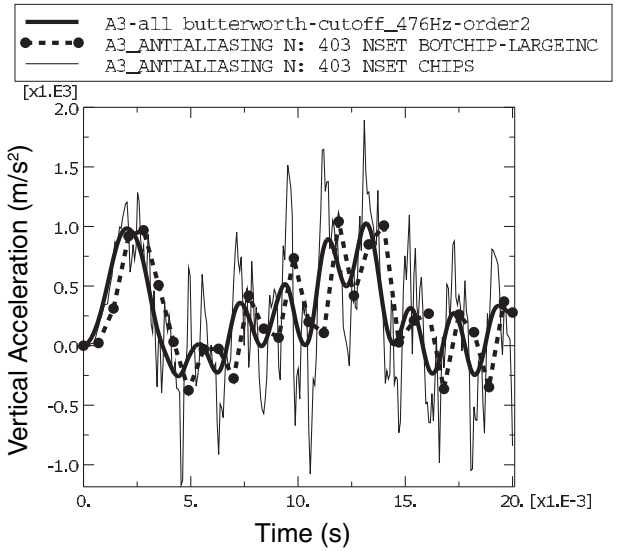


Figure 12–64 Comparison of acceleration filtered in Abaqus/Explicit and Abaqus/Viewer.

To develop a better understanding of the Abaqus/Viewer filtering capabilities, return to the **Operate on XY Data** dialog box and filter the acceleration data with other filter options. For example, try different cutoff frequencies.

Can you confirm that the cutoff frequency of 2.4 kHz associated with the built-in anti-aliasing filter with a time increment size of 0.07 was appropriate? Does increasing the cutoff frequency to 6 kHz, 7 kHz, or even 10 kHz produce significantly different results?

You should find that a moderate increase in the cutoff frequency does not have a significant effect on the results, implying that we probably have not missed physically meaningful frequency content when we filtered with a cutoff frequency of 2.4 kHz.

Compare the results of filtering the acceleration data with Butterworth and Chebyshev Type I filters. The Chebyshev filter requires a ripple factor parameter (*rippleFactor*), which indicates how much oscillation you will allow in exchange for an improved filter response; see "Filtering output and operating on output in Abaqus/Explicit" in "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual for more information. For the Chebyshev Type I filter a ripple factor of 0.071 will result in a very flat pass band with a ripple that is only 0.5%.

You may not notice much difference between the filters when the cutoff frequency is 5 kHz, but what about when the cutoff frequency is 2 kHz? What happens when you increase the order of the Chebyshev Type I *filter?*

Compare your results to those shown in Figure 12–65.

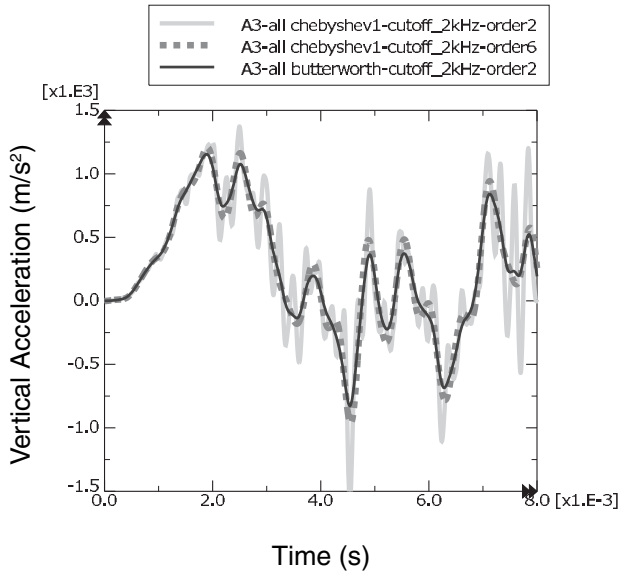


Figure 12–65 Comparison of acceleration filtered with Butterworth and Chebyshev Type I filters.

Note: The Abaqus/Viewer postprocessing filters are second-order by default. To define a higher order filter you can use the *filterOrder* parameter with the **butterworthFilter** and the **chebyshev1Filter** operators. For example, use the following expression in the **Operate on XY Data** dialog box to filter `A3-all` with a sixth-order Chebyshev Type I filter using a cutoff frequency of 2 kHz and a ripple factor of 0.017.

```
chebyshev1Filter ( xyData="A3-all" , cutoffFrequency=2000,
 rippleFactor= 0.017, filterOrder=6)
```

The second-order Chebyshev Type I filter with a ripple factor of 0.071 is a relatively weak filter, so some of the frequency content above the 2 kHz cutoff frequency is not filtered out. When the filter order is increased, the filter response is improved so that the results are more like the equivalent Butterworth filter. For more information on the *X–Y* data filters available in Abaqus/Viewer see "Operating on saved *X–Y* data objects," Section 47.4 of the Abaqus/CAE User's Manual.

Filtering strain history in Abaqus/Viewer

Strain in the circuit board near the location of the chips is another result that may assist us in determining the effectiveness of the foam packaging. If the strain under the chips exceeds a limiting value, the solder securing the chips to the board will fail. We wish to identify the peak strain in

any direction. Therefore, the maximum and minimum principal logarithmic strains are of interest. Principal strains are one of a number of Abaqus results that are derived from nonlinear operators; in this case a nonlinear function is used to calculate principal strains from the individual strain components. Some other common results that are derived from nonlinear operators are principal stresses, Mises stress, and equivalent plastic strains. Care must be taken when filtering results that are derived from nonlinear operators, because nonlinear operators (unlike linear ones) can modify the frequency of the original result. Filtering such a result may have undesirable consequences; for example, if you remove a portion of the frequency content that was introduced by the application of the nonlinear operator, the filtered result will be a distorted representation of the derived quantity. In general, you should either avoid filtering quantities derived from nonlinear operators or filter the underlying quantities before calculating the derived quantity using the nonlinear operator.

The strain history output for this analysis was recorded every 0.07 ms using the built-in anti-aliasing filter. To verify that the anti-aliasing filter did not distort the principal strain results, we will calculate the principal logarithmic strains using the filtered strain components and compare the result to the filtered principal logarithmic strains.

To calculate the principal logarithmic strains:

1. In the Results Tree, filter the **History Output** according to **`*LE*`**, select the logarithmic strain component **`LE11`** on the **`SPOS`** surface of the element in set **`BOTPART`**, and save the data as **`LE11`**.
2. Similarly, save the **`LE12`** and **`LE22`** strain components for the same element as **`LE12`** and **`LE22`**, respectively.
3. In the Results Tree, double-click **XYData**; then select **Operate on XY data** in the **Create XY Data** dialog box. Click **Continue**.
4. In the **Operate on XY Data** dialog box, use the saved logarithmic strain components to calculate the maximum principal logarithmic strain. The expression at the top of the dialog box should appear as:

   ```
   (("LE11"+"LE22")/2) + sqrt( power(("LE11"-"LE22")/2,2)
    + power("LE12"/2,2) )
   ```

5. Click **Save As** to save calculated maximum principal logarithmic strain as **`LEP-Max`**.
6. Edit the expression in the **Operate on XY Data** dialog box to calculate the minimum principal logarithmic strain. The modified expression should appear as:

   ```
   (("LE11"+"LE22")/2) - sqrt( power(("LE11"-"LE22")/2,2)
    + power("LE12"/2,2) )
   ```

7. Click **Save As** to save calculated minimum principal logarithmic strain as **`LEP-Min`**.

   In order to plot the calculated principal logarithmic strains with the same *Y*-axis as the strains recorded during the analysis, change the *Y*-value type to strain.

8. In the **XYData** container of the Results Tree, click mouse button 3 on **LEP-Max**; and select **Edit** from the menu that appears.
9. In the **Edit XY Data** dialog box, choose **Strain** as the *Y*-value type.
10. Similarly, edit **LEP-Min** and select **Strain** as the *Y*-value type.
11. Using the Results Tree, plot **LEP-Man** and **LEP-Min** along with the principal strains recorded during the analysis (**LEP1** and **LEP2**) for the element in set **BOTPART**.
12. As before, customize the plot appearance to obtain a plot similar to Figure 12–66.

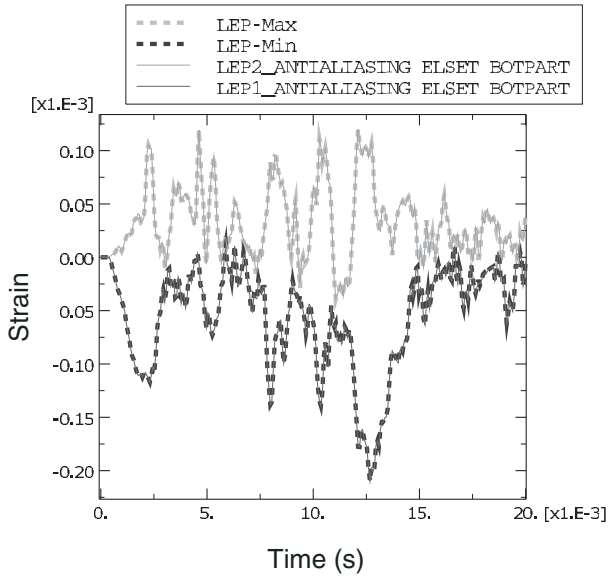


Figure 12–66 Principal logarithmic strain values versus time.

In Figure 12–66 we see that the filtered principal logarithmic strain curves recorded during the analysis are indistinguishable from the principal logarithmic strain curves calculated from the filtered strain components. Therefore the anti-aliasing filter (cutoff frequency 2.4 kHz) did not remove any of the frequency content introduced by the nonlinear operation to calculate principal strains form the original strain data. Next, filter the strain data with a lower cutoff frequency of 500 Hz.

To filter principal logarithmic strains with a cutoff frequency of 500 Hz:

1. In the Results Tree, double-click **XYData**; then select **Operate on XY data** in the **Create XY Data** dialog box. Click **Continue**.

2. In the **Operate on XY Data** dialog box, filter the maximum principal logarithmic strain **LEP-Max** using a second-order Butterworth filter with a cutoff frequency of 500 Hz. The expression at the top of the dialog box should appear as:

```
butterworthFilter(xyData="LEP-Max", cutoffFrequency=500)
```

3. Click **Save As** to save the calculated maximum principal logarithmic strain as **LEP-Max-FilterAfterCalc-bw500**.
4. Similarly, filter the logarithmic strain components **LE11**, **LE12**, and **LE22** using the same second-order Butterworth filter with a cutoff frequency of 500 Hz. Save the resulting curves as **LE11-bw500**, **LE12-bw500**, and **LE22-bw500**, respectively.
5. Now calculate the maximum principal logarithmic strain using the filtered logarithmic strain components. The expression at the top of the **Operate on XY Data** dialog box should appear as:

```
(("LE11-bw500"+"LE22-bw500")/2) + sqrt(
 power(("LE11-bw500"-"LE22-bw500")/2,2) +
 power("LE12-bw500"/2,2) )
```

6. Click **Save As** to save the calculated maximum principal logarithmic strain as **LEP-Max-CalcAfterFilter-bw500**.
7. In the **XYData** container of the Results Tree, click mouse button 3 on **LEP-Max-CalcAfterFilter-bw500**; and select **Edit** from the menu that appears.
8. In the **Edit XY Data** dialog box, choose **Strain** as the *Y*-value type.
9. Plot **LEP-Max-CalcAfterFilter-bw500** and **LEP-Max-FilterAfterCalc-bw500** as shown in Figure 12–67.

In Figure 12–67 you can see that there is a significant difference between filtering the strain data before and after the principal strain calculation. The curve that was filtered after the principal strain calculation is distorted because some of the frequency content introduced by applying the nonlinear principal-stress operator is higher than the 500 Hz filter cutoff frequency. In general, you should avoid directly filtering quantities that have been derived from nonlinear operators; whenever possible filter the underlying components and then apply the nonlinear operator to the filtered components to calculate the desired derived quantity.

Strategy for recording and filtering Abaqus/Explicit history output

Recording output for every increment in Abaqus/Explicit generally produces much more data than you need. The real-time filtering capability allows you to request history output less frequently without distorting the results due to aliasing. However, you should ensure that your output rate and filtering choices have not removed physically meaningful frequency content nor distorted the results (for example, by introducing a large time delay or by removing frequency content introduced by nonlinear operators). Keep in mind that no amount of postprocessing filtering can recover frequency content filtered out during the analysis, nor can postprocessing filtering recover an original signal

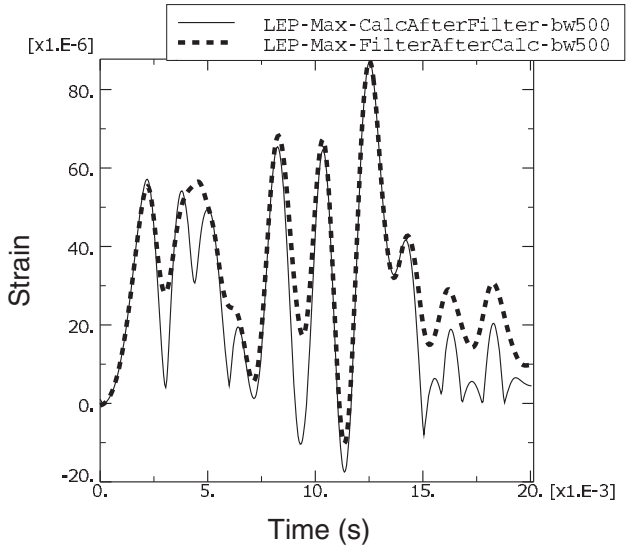


Figure 12–67 Principal logarithmic strain calculated before and after filtering (cutoff frequency 500 Hz).

from aliased data. In addition, it may not be obvious when results have been over-filtered or aliased if additional data are not available for comparison. A good strategy is to choose a relatively high output rate and use the Abaqus/Explicit filters to prevent aliasing of the history output, so that valid and rich results are written to the output database. You may even wish to request output at every increment for a couple of critical locations. After the analysis completes, use the postprocessing tools in Abaqus/Viewer to quickly and iteratively apply additional filtering as desired.

12.11 Compatibility between Abaqus/Standard and Abaqus/Explicit

There are fundamental differences in the mechanical contact algorithms in Abaqus/Standard and Abaqus/Explicit. These differences are reflected in how contact conditions are defined. The main differences are the following:

- For contact pairs Abaqus/Standard typically uses a pure master-slave relationship for the contact constraints by default (see “Defining contact pairs in Abaqus/Standard,” Section 35.3.1 of the Abaqus Analysis User’s Manual); the nodes of the slave surface are constrained not to penetrate into the master surface. The nodes of the master surface can, in principle, penetrate into the slave surface. Abaqus/Explicit includes this formulation but typically uses a balanced

master-slave weighting by default (see "Contact formulations for contact pairs in Abaqus/Explicit," Section 37.2.2 of the Abaqus Analysis User's Manual).

- The contact formulations in Abaqus/Standard and Abaqus/Explicit differ in many respects. For example, Abaqus/Standard provides a surface-to-surface formulation, while Abaqus/Explicit provides an edge-to-edge formulation.
- The constraint enforcement methods in Abaqus/Standard and Abaqus/Explicit differ in some respects. For example, both Abaqus/Standard and Abaqus/Explicit provide penalty constraint methods, but the default penalty stiffnesses differ.
- Abaqus/Standard and Abaqus/Explicit both provide a small-sliding contact formulation (see "Contact formulations in Abaqus/Standard," Section 37.1.1 of the Abaqus Analysis User's Manual, and "Contact formulations for contact pairs in Abaqus/Explicit," Section 37.2.2 of the Abaqus Analysis User's Manual). However, the small-sliding contact formulation in Abaqus/Standard transfers the load to the master nodes according to the current position of the slave node. Abaqus/Explicit always transfers the load through the anchor point.

As a result of these differences, contact definitions specified in an Abaqus/Standard analysis cannot be imported into an Abaqus/Explicit analysis and vice versa (see "Transferring results between Abaqus/Explicit and Abaqus/Standard," Section 9.2.2 of the Abaqus Analysis User's Manual).

12.12 Related Abaqus examples

- "Indentation of a crushable foam plate," Section 3.2.10 of the Abaqus Benchmarks Manual
- "Pressure penetration analysis of an air duct kiss seal," Section 1.1.16 of the Abaqus Example Problems Manual
- "Deep drawing of a cylindrical cup," Section 1.3.4 of the Abaqus Example Problems Manual

12.13 Suggested reading

The following references provide additional information on contact analysis with finite element methods. They allow the interested user to explore the topic in more depth.

General texts on contact analysis

- Belytschko, T., W. K. Liu, and B. Moran, *Nonlinear Finite Elements for Continua and Structures*, Wiley & Sons, 2000.
- Crisfield, M. A., *Non-linear Finite Element Analysis of Solids and Structures, Volume II: Advanced Topics*, Wiley & Sons, 1997.
- Johnson, K. L., *Contact Mechanics*, Cambridge, 1985.
- Oden, J. T., and G. F. Carey, *Finite Elements: Special Problems in Solid Mechanics,* Prentice-Hall, 1984.

General text on digital signal proccesing

- Stearns, S. D., and R. A. David, *Signal Processing Algorithms in MATLAB*, Prentice Hall P T R, 1996.

12.14 Summary

- Contact analyses require a careful, logical approach. Divide the analysis into several steps if necessary, and apply the loading slowly making sure that the contact conditions are well established.
- In general, it is best to use a separate step for each part of the analysis in Abaqus/Standard even if it is just to change boundary conditions to loads. You will almost certainly end up with more steps than anticipated, but the model should converge much more easily. Contact analyses are much more difficult to complete if you try to apply all the loads in one step.
- In Abaqus/Standard achieve stable contact conditions between all components before applying the working loads to the structure. If necessary, apply temporary boundary conditions, which may be removed at a later stage. The final results should be unaffected, provided that the constraints produce no permanent deformation.
- Do not apply boundary conditions to nodes on contact surfaces that constrain the node in the direction of contact in Abaqus/Standard. If there is friction, do not constrain these nodes in any degree of freedom: zero pivot messages may result.
- Always try to use first-order elements for contact simulations in Abaqus/Standard.
- Both Abaqus/Standard and Abaqus/Explicit provide two distinct algorithms for modeling contact: general contact (defined with the ∗CONTACT option) and contact pairs (defined with the ∗CONTACT PAIR option).
- General contact interactions allow you to define contact between many or all regions of a model; contact pair interactions describe contact between two surfaces or between a single surface and itself.
- General contact can be defined in the model or history part of the input file; contact pairs are defined in the history part of the input file.
- Surfaces used with the general contact algorithm can span multiple unattached bodies. More than two surface facets can share a common edge. In contrast, all surfaces used with the contact pair algorithm must be continuous and simply connected.
- Surfaces are defined using the ∗SURFACE option. Individual nodes can be included in a contact pair by using the TYPE=NODE parameter on the ∗SURFACE option. Analytical rigid surfaces are assigned to a rigid body by using the ANALYTICAL SURFACE parameter on the ∗RIGID BODY option. The parameters TYPE=SEGMENTS, CYLINDRICAL, or REVOLUTION on the ∗SURFACE option specify the type of analytical rigid surface.

- In Abaqus/Explicit single-sided surfaces on shell, membrane, or rigid elements must be defined so that the normal directions do not “flip” as the surface is traversed.
- Abaqus/Explicit does not smooth rigid surfaces; they are faceted like the underlying elements. Coarse meshing of discrete rigid surfaces can produce noisy solutions with the contact pair algorithm. The general contact algorithm does include some numerical rounding of features.
- Tie constraints are a useful means of mesh refinement in Abaqus.
- Abaqus/Explicit adjusts the nodal coordinates without strain to remove any initial overclosures prior to the first step. If the adjustments are large with respect to the element dimensions, elements can become severely distorted.
- In subsequent steps any nodal adjustments to remove initial overclosures in Abaqus/Explicit induce strains that can potentially cause severe mesh distortions.
- When you are interested in results that are likely to contain high frequency oscillations, such as accelerations in an impact problem, request Abaqus/Explicit history output with a relatively high output rate and (if the output rate is less than every increment) apply an anti-aliasing filter; then, use a postprocessing filter if stronger filtering is desired.
- The Abaqus Analysis User’s Manual contains more detailed discussions of contact modeling in Abaqus. “Contact interaction analysis: overview,” Section 35.1.1 of the Abaqus Analysis User’s Manual, is a good place to begin further reading on the subject.

13. Quasi-Static Analysis with Abaqus/Explicit

The explicit solution method is a true dynamic procedure originally developed to model high-speed impact events in which inertia plays a dominant role in the solution. Out-of-balance forces are propagated as stress waves between neighboring elements while solving for a state of dynamic equilibrium. Since the minimum stable time increment is usually quite small, most problems require a large number of increments.

The explicit solution method has proven valuable in solving quasi-static problems as well—Abaqus/Explicit solves certain types of static problems more readily than Abaqus/Standard does. One advantage of the explicit procedure over the implicit procedure is the greater ease with which it resolves complicated contact problems. In addition, as models become very large, the explicit procedure requires fewer system resources than the implicit procedure. Refer to "Comparison of implicit and explicit procedures," Section 2.4, for a detailed comparison of the implicit and explicit procedures.

Applying the explicit dynamic procedure to quasi-static problems requires some special considerations. Since a static solution is, by definition, a long-time solution, it is often computationally impractical to simulate an event in its natural time scale, which would require an excessive number of small time increments. To obtain an economical solution, the event must be accelerated in some way. The problem is that as the event is accelerated, the state of static equilibrium evolves into a state of dynamic equilibrium in which inertial forces become more dominant. The goal is to model the process in the shortest time period in which inertial forces remain insignificant.

Quasi-static analyses can also be conducted in Abaqus/Standard. Quasi-static stress analysis in Abaqus/Standard is used to analyze linear or nonlinear problems with time-dependent material response (creep, swelling, viscoelasticity, and two-layer viscoplasticity) when inertia effects can be neglected. For more information on quasi-static analysis in Abaqus/Standard, see "Quasi-static analysis," Section 6.2.5 of the Abaqus Analysis User's Manual.

13.1 Analogy for explicit dynamics

To provide you with a more intuitive understanding of the differences between a slow, quasi-static loading case and a rapid loading case, we use the analogy illustrated in Figure 13–1. The figure shows two cases of an elevator full of passengers. In the slow case the door opens and you walk in. To make room, the occupants adjacent to the door slowly push their neighbors, who push their neighbors, and so on. This disturbance passes through the elevator until the people next to the walls indicate that they cannot move. A series of waves pass through the elevator until everyone has reached a new equilibrium position. If you increase your speed slightly, you will shove your neighbors more forcefully than before, but in the end everyone will end up in the same position as in the slow case.

In the fast case the door opens and you run into the elevator at high speed, permitting the occupants no time to rearrange themselves to accommodate you. You will injure the two people directly in front of the door, while the other occupants will be unaffected.

Figure 13–1 Analogy for slow and fast loading cases.

The same thinking is true for quasi-static analyses. The speed of the analysis often can be increased substantially without severely degrading the quality of the quasi-static solution; the end result of the slow case and a somewhat accelerated case are nearly the same. However, if the analysis speed is increased to a point at which inertial effects dominate, the solution tends to localize, and the results are quite different from the quasi-static solution.

13.2 Loading rates

The actual time taken for a physical process is called its natural time. Generally, it is safe to assume that performing an analysis in the natural time for a quasi-static process will produce accurate static results. After all, if the real-life event actually occurs in a natural time scale in which velocities are zero at the conclusion, a dynamic analysis should be able to capture the fact that the analysis has, in fact, achieved a steady state. You can increase the loading rate so that the same physical event occurs in less time as long as the solution remains nearly the same as the true static solution and dynamic effects remain insignificant.

13.2.1 Smooth amplitude curves

For accuracy and efficiency quasi-static analyses require the application of loading that is as smooth as possible. Sudden, jerky movements cause stress waves, which can induce noisy or inaccurate solutions. Applying the load in the smoothest possible manner requires that the acceleration changes only a small amount from one increment to the next. If the acceleration is smooth, it follows that the changes in velocity and displacement are also smooth.

Abaqus has a simple, built-in type of amplitude called SMOOTH STEP that automatically creates a smooth loading amplitude. When you define time-amplitude data pairs using ∗AMPLITUDE, DEFINITION=SMOOTH STEP, Abaqus/Explicit automatically connects each of your data pairs with curves whose first and second derivatives are smooth and whose slopes are zero at each of your data points. Since both of these derivatives are smooth, you can apply a displacement loading with SMOOTH STEP using only the initial and final data points, and the intervening motion will be smooth.

Using this type of loading amplitude allows you to perform a quasi-static analysis without generating waves due to discontinuity in the rate of applied loading. For example, for the following amplitude definition Abaqus/Explicit creates the amplitude curve shown in Figure 13–2:

```
*AMPLITUDE, DEFINITION=SMOOTH STEP
0.0, 0.0, 1.0E-5, 1.0
```

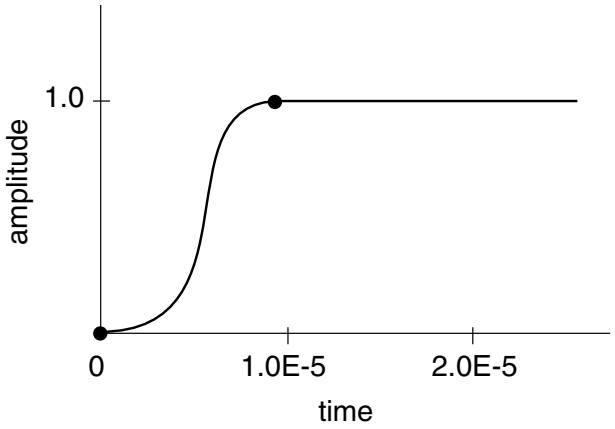


Figure 13–2 Amplitude definition using ∗AMPLITUDE, DEFINITION=SMOOTH STEP.

13.2.2 Structural problems

In a static analysis the lowest mode of the structure usually dominates the response. Knowing the frequency and, correspondingly, the period of the lowest mode, you can estimate the time required to obtain the proper static response. To illustrate the problem of determining the proper loading rate, consider the deformation of a side intrusion beam in a car door by a rigid cylinder, as shown in Figure 13–3. The actual test is quasi-static.

The response of the beam varies greatly with the loading rate. At an extremely high impact velocity of 400 m/s, the deformation in the beam is highly localized, as shown in Figure 13–4. To obtain a better quasi-static solution, consider the lowest mode.

The frequency of the lowest mode is approximately 250 Hz, which corresponds to a period of 4 milliseconds. The natural frequencies can easily be calculated using the ∗FREQUENCY procedure in Abaqus/Standard. To deform the beam by the desired 0.2 m in 4 milliseconds, the velocity of the cylinder is 50 m/s. While 50 m/s still seems like a high, impact velocity, the inertial forces become secondary to the overall stiffness of the structure, and the deformed shape—shown in Figure 13–5—indicates a much better quasi-static response.

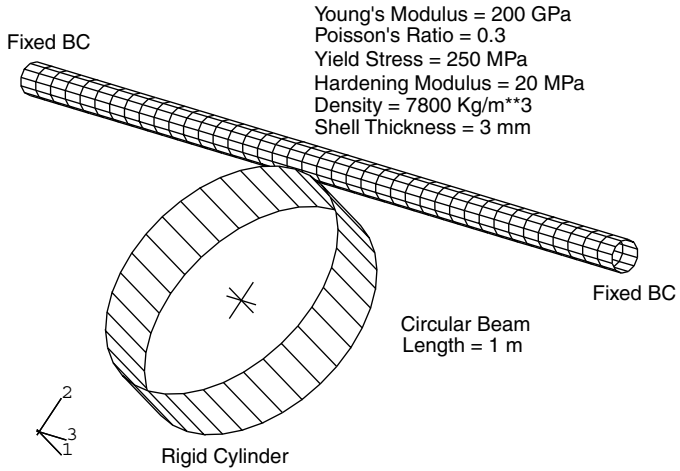


Figure 13–3 Rigid cylinder impacting beam.

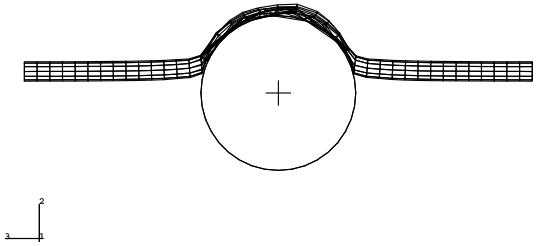


Figure 13–4 Impact velocity of 400 m/s.

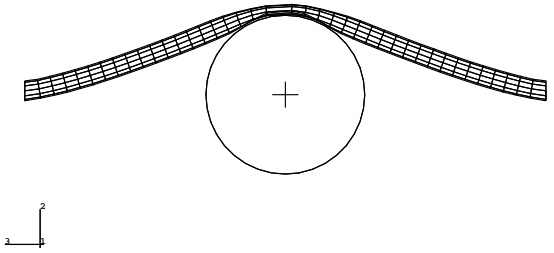


Figure 13–5 Impact velocity of 50 m/s.

While the overall structural response appears to be what we expect as a quasi-static solution, it is usually desirable to increase the loading time to 10 times the period of the lowest mode to be certain that the solution is truly quasi-static. To improve the results even further, the velocity of the rigid cylinder could be ramped up gradually—for example, using a SMOOTH STEP amplitude definition—thereby easing the initial impact.

13.2.3 Metal forming problems

Artificially increasing the speed of forming events is necessary to obtain an economical solution, but how large a speedup can we impose and still obtain an acceptable static solution? If the deformation of the sheet metal blank corresponds to the deformed shape of the lowest mode, the time period of the lowest structural mode can be used as a guideline for forming speed. However, in forming processes the rigid dies and punches can constrain the blank in such a way that its deformation may not relate to the structural modes. In such cases a general recommendation is to limit punch speeds to less than 1% of the sheet metal wave speed. For typical processes the punch speed is on the order of 1 m/s, while the wave speed of steel is approximately 5000 m/s. This recommendation, therefore, suggests a factor of 50 as an upper bound on the speedup of the punch velocity.

The suggested approach to determining an acceptable punch velocity involves running a series of analyses at various punch speeds in the range of 3 to 50 m/s. Perform the analyses in the order of fastest to slowest since the solution time is inversely proportional to the punch velocity. Examine the results of the analyses, and get a feel for how the deformed shapes, stresses, and strains vary with punch speed. Some indications of excessive punch speeds are unrealistic, localized stretching and thinning as well as the suppression of wrinkling. If you begin with a punch speed of, for example, 50 m/s, and decrease it from there, at some point the solutions will become similar from one punch speed to the next—an indication that the solutions are converging on a quasi-static solution. As inertial effects become less significant, differences in simulation results also become less significant.

As the loading rate is increased artificially, it becomes more and more important to apply the loads in a gradual and smooth manner. For example, the simplest way to load the punch is to impose a constant velocity throughout the forming step. Such a loading causes a sudden impact load onto the sheet metal blank at the start of the analysis, which propagates stress waves through the blank and may produce undesired results. The effect of any impact load on the results becomes more pronounced as the loading rate is increased. Ramping up the punch velocity from zero using a smooth step amplitude curve minimizes these adverse effects.

Springback

Springback is often an important part of a forming analysis because the springback analysis determines the shape of the final, unloaded part. While Abaqus/Explicit is well-suited for forming simulations, springback poses some special difficulties. The main problem with performing springback simulations within Abaqus/Explicit is the amount of time required to obtain a steady-state solution. Typically, the loads must be removed very carefully, and damping must

be introduced to make the solution time reasonable. Fortunately, the close relationship between Abaqus/Explicit and Abaqus/Standard allows a much more efficient approach.

Since springback involves no contact and usually includes only mild nonlinearities, Abaqus/Standard can solve springback problems much faster than Abaqus/Explicit can. Therefore, the preferred approach to springback analyses is to import the completed forming model from Abaqus/Explicit into Abaqus/Standard. You must create an Abaqus/Standard input file that will import the forming results and perform the springback analysis. Using the *IMPORT option within the Abaqus/Standard input file, you specify the element sets that you wish to import. Usually, the entire deformable mesh is imported. The nodes, elements, and section properties are imported automatically, but you must redefine the materials and boundary conditions. Once the springback analysis is complete, you can import the model back into Abaqus/Explicit to continue with another forming stage.

13.3 Mass scaling

Mass scaling enables an analysis to be performed economically without artificially increasing the loading rate. Mass scaling is the only option for reducing the solution time in simulations involving a rate-dependent material or rate-dependent damping, such as dashpots. In such simulations increasing the loading rate is not an option because material strain rates increase by the same factor as the loading rate. When the properties of the model change with the strain rate, artificially increasing the loading rate artificially changes the process.

The following equations show how the stable time increment is related to the material density. As discussed in "Definition of the stability limit," Section 9.3.2, the stability limit for the model is the minimum stable time increment of all elements. It can be expressed as

$$\Delta t = \frac{L^e}{c_d},$$

where L^e is the characteristic element length and c_d is the dilatational wave speed of the material. The dilatational wave speed for a linear elastic material with Poisson's ratio equal to zero is given by

$$c_d = \sqrt{\frac{E}{\rho}},$$

where ρ is the material density.

According to the above equations, artificially increasing the material density, ρ, by a factor of f^2 decreases the wave speed by a factor of f and increases the stable time increment by a factor of f. Remember that when the global stability limit is increased, fewer increments are required to perform the same analysis, which is the goal of mass scaling. Scaling the mass, however, has exactly the same influence on inertial effects as artificially increasing the loading rate. Therefore, excessive mass scaling, just like excessive loading rates, can lead to erroneous solutions. The suggested approach to determining

an acceptable mass scaling factor, then, is similar to the approach to determining an acceptable loading rate scaling factor. The only difference to the approach is that the speedup associated with mass scaling is the square root of the mass scaling factor, whereas the speedup associated with loading rate scaling is proportional to the loading rate scaling factor. For example, a mass scaling factor of 100 corresponds exactly to a loading rate scaling factor of 10.

There are several ways to implement mass scaling in your model using the ∗FIXED MASS SCALING or ∗VARIABLE MASS SCALING option in the history definition of the input file. Since the option is part of the history definition, it can be changed from step to step, allowing great flexibility. Refer to "Mass scaling," Section 11.6.1 of the Abaqus Analysis User's Manual, for details.

13.4 Energy balance

The most general means of evaluating whether or not a simulation is producing an appropriate quasi-static response involves studying the various model energies. The following is the energy balance equation in Abaqus/Explicit:

$$E_I + E_V + E_{FD} + E_{KE} + E_{IHE} - E_W - E_{PW} - E_{CW} - E_{MW} - E_{HF} = E_{total} = \text{constant},$$

where E_I is the internal energy, E_V is the viscous energy dissipated, E_{FD} is the frictional energy dissipated, E_{KE} is the kinetic energy, E_{IHE} is the internal heat energy, E_W is the work done by the externally applied loads, and E_{PW}, E_{CW}, and E_{MW} are the work done by contact penalties, by constraint penalties, and by propelling added mass, respectively. E_{HF} is the external heat energy through external fluxes. The sum of these energy components is E_{total}, which should be constant. In the numerical model E_{total} is only approximately constant, generally with an error of less than 1%.

To illustrate energy balance with a simple example, consider the uniaxial tensile test shown in Figure 13–6. The energy history for the quasi-static test would appear as shown in Figure 13–7. If a simulation is quasi-static, the work applied by the external forces is nearly equal to the internal energy of the system. The viscously dissipated energy is generally small unless viscoelastic materials, discrete dashpots, or material damping are used. We have already established that the inertial forces are negligible in a quasi-static analysis because the velocity of the material in the model is very small. The corollary to both of these conditions is that the kinetic energy is also small. As a general rule the kinetic energy of the deforming material should not exceed a small fraction (typically 5% to 10%) of its internal energy throughout most of the process.

When comparing the energies, remember that Abaqus/Explicit reports a global energy balance, which includes the kinetic energy of any rigid bodies with mass. Since only the deformable bodies are of interest when evaluating the results, the kinetic energy of the rigid bodies should be subtracted from E_{total} when evaluating the energy balance.

For example, if you are simulating a transport problem with rolling rigid dies, the kinetic energy of the rigid bodies may be a significant portion of the total kinetic energy of the model. In such cases you must subtract the kinetic energy associated with rigid body motions before a meaningful comparison with internal energy can be made.

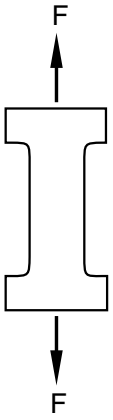


Figure 13–6 Uniaxial tensile test.

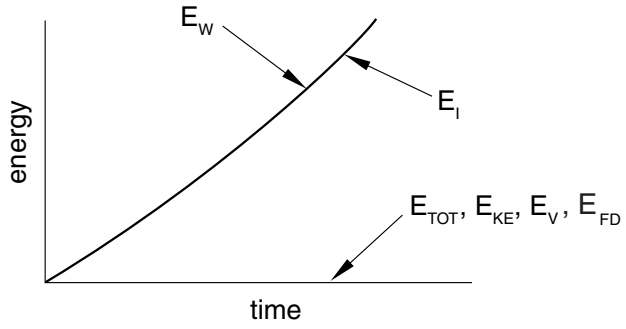


Figure 13–7 Energy history for quasi-static tensile test.

13.5 Example: forming a channel in Abaqus/Explicit

In this example you will solve the channel forming problem from Chapter 12, "Contact," using Abaqus/Explicit. You will then compare the results from the Abaqus/Standard and Abaqus/Explicit analyses.

You will make modifications to the model created for the Abaqus/Standard analysis so that you are able to run it in Abaqus/Explicit. These modifications include adding density to the material model and changing the steps. Before running the Abaqus/Explicit analysis, you will use the frequency extraction procedure in Abaqus/Standard to determine the time period required to obtain a proper quasi-static response.

13.5.1 Preprocessing—rerunning the model with Abaqus/Explicit

Before starting, save a copy of **`channel.inp`** as **`channel_freq.inp`**. Make all subsequent changes to the **`channel_freq.inp`** file. **`channel_freq.inp`** is also available in the "Forming a channel with Abaqus/Standard," Section A.13. In this section you will modify the input file in order to perform a frequency extraction of the blank using Abaqus/Standard.

Determining an appropriate step time

"Loading rates," Section 13.2, discusses the procedures for determining the appropriate step time for a quasi-static process. We can determine an approximate lower bound on step time duration if we know the lowest natural frequency, the *fundamental* frequency, of the blank. One way to obtain such information is to run a frequency analysis in Abaqus/Standard. In this forming analysis the punch deforms the blank into a shape similar to the lowest mode. Therefore, it is important that the time for the forming stage is greater than or equal to the time period for the lowest mode if you wish to model structural, as opposed to localized, deformation.

To perform a natural frequency extraction:

1. Modify the *MATERIAL option block to include the *DENSITY suboption:

   ```
   *DENSITY
   7800.,
   ```

2. Delete all data associated with the die, holder, and punch (*SURFACE, *RIGID BODY, *CONTACT PAIR, etc.). These rigid parts are not necessary for the frequency analysis.

3. Delete all steps except for the first one. Change the procedure type to *FREQUENCY, and change its step description to **`Frequency modes`**. Request five eigenvalues using the default Lanczos eigensolver.

4. Delete all boundary conditions except the boundary condition applied to the set **`CENTER`**. This leaves the blank constrained with a symmetry boundary condition applied to the left end.

   The revised history data appears below.

   ```
   *STEP, PERTURBATION
   Frequency modes
   *FREQUENCY
   5,
   *BOUNDARY
   CENTER, XSYMM
   *END STEP
   ```

Note: Since the frequency extraction step is a linear perturbation procedure, nonlinear material properties will be ignored. In this analysis the left end of the blank is constrained in the *x*-direction and cannot rotate about the normal; however, it is not constrained in the *y*-direction. Therefore, the first mode extracted will be a rigid body mode. The frequency of the second mode will determine the appropriate time period for the quasi-static analysis in Abaqus/Explicit.

5. Save the **`channel_freq.inp`** input file, submit the job for analysis, and monitor the solution progress.
6. When the analysis is complete, enter Abaqus/Viewer and open the output database file created by this analysis. From the main menu bar, select **Plot→Deformed Shape**; or use the tool in the toolbox.

The deformed model shape for the first vibration mode is plotted (it is a rigid body mode). Advance the plot to the second mode of the blank. Superimpose the undeformed model shape on the deformed model shape.

The frequency analysis shows that the blank has a fundamental frequency of 140 Hz, corresponding to a period of 0.00714 s. Figure 13–8 shows the displaced shape of the second mode. We now know that the shortest step time for the forming analysis is 0.00714 s.

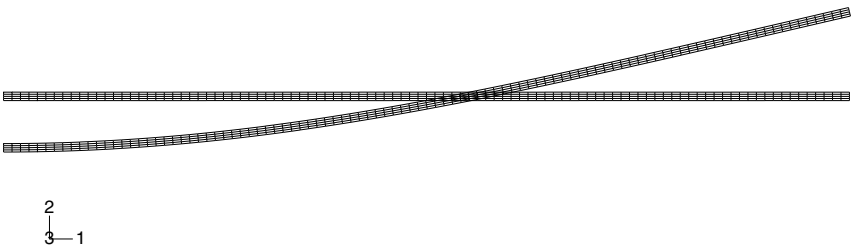


Figure 13–8 Second mode of the blank from the Abaqus/Standard frequency analysis.

Creating the Abaqus/Explicit forming analysis

The goal of the forming process is to quasi-statically form a channel with a punch displacement of 0.03 m. In selecting loading rates for quasi-static analyses, it is recommended that you begin with faster loading rates and decrease the loading rates as necessary to converge on a quasi-static solution more quickly. However, if you wish to increase the likelihood of a quasi-static result in your first analysis attempt, you should consider step times that are a factor of 10 to 50 times slower than that corresponding to the fundamental frequency. In this analysis you will start with a time period of 0.007 s for the forming analysis step, which is based on the frequency analysis

performed in Abaqus/Standard, which shows that the blank has a fundamental frequency of 140 Hz, corresponding to a time period of 0.00714 s. This time period corresponds to a constant punch velocity of 4.3 m/s. You will examine the kinetic and internal energy results carefully to check that the solution does not include significant dynamic effects.

Before starting, save a copy of **channel.inp** as **channel_xpl.inp**. Make all subsequent changes to the **channel_xpl.inp** file. **channel_xpl.inp** is also available in "Forming a channel with Abaqus/Standard," Section A.13. In this section you will modify the input file in order to perform the forming analysis of the blank using Abaqus/Explicit. This analysis also requires a density setting for the material model **Steel**, so repeat the density specification step in "Example: forming a channel in Abaqus/Explicit," Section 13.5.

A concentrated force will be applied to the blank holder. To compute the dynamic response of the holder, a point mass must be assigned to its rigid body reference point. The actual mass of the holder is not important; what is important is that the mass should be of the same order of magnitude as the mass of the blank (0.78 kg) to minimize noise in the contact calculations. Choose a point mass value of 0.1 kg. To assign the mass, append the following statements to the option block for the rigid blank holder.

```
*ELEMENT, TYPE=MASS, ELSET=HOLDER_MASS
8000, 8000
*MASS, ELSET=HOLDER_MASS
0.1,
```

For the first attempt of this metal forming analysis, you will use tabular amplitude curves with the default smoothing parameter for both the application of the holder force and the punch stroke. Create a tabular amplitude curve for application of the holder force named **Ramp1** using the data in Table 13–1. Define a second tabular amplitude curve for the punch stroke named **Ramp2** using the data in Table 13–2.

Table 13–1 Ramp amplitude data for **Ramp1**.

Time (sec)	Amplitude
0.0	0.0
0.0001	1.0

Table 13–2 Ramp amplitude data for **Ramp2**.

Time (sec)	Amplitude
0.0	0.0
0.007	1.0

The ramp amplitude data are defined in *AMPLITUDE statements, as shown below.

```
*AMPLITUDE, NAME=RAMP1
0., 0., 0.0001, 1.
*AMPLITUDE, NAME=RAMP2
0., 0., 0.007, 1.
```

As with the Abaqus/Standard analysis, you will need two steps for the Abaqus/Explicit analysis. In the first step the holder force is applied; in the second step the punch stroke is applied. This can be easily done by modifying the existing steps. For each step, replace the step procedure with the explicit dynamics procedure. For the first step, specify a time period of **0.0001** s. This time period is appropriate for the application of the holder force because it is long enough to avoid dynamic effects but short enough to prevent a significant impact on the run time for the job. Move the contact pair definitions from the model data to the first step definition (but delete the TYPE=SURFACE to SURFACE parameter for each contact pair as it is relevant only for Abaqus/Standard). In the step, retain the boundary conditions and concentrated force definitions. The amplitude curve associated with the concentrated force should be set to **RAMP1**. Modify the history output request for the punch reference node to request output at 200 evenly spaced intervals using the built-in anti-aliasing filter.

After these changes the first step definition will appear in the input file as follows:

```
**
** Step 1
**
*STEP
Apply holder force
*DYNAMIC, EXPLICIT
, 0.0001
*CONTACT PAIR, INTERACTION=FRIC
BLANK_B, DIE
BLANK_T, HOLDER
*CONTACT PAIR, INTERACTION=NOFRIC
BLANK_T, PUNCH
*BOUNDARY
CENTER  , XSYMM
REFDIE  , 1, 6
REFPUNCH, 1, 6
REFHOLD , 1, 1
REFHOLD , 6, 6
*CLOAD, AMPLITUDE=RAMP1
REFHOLD, 2, -4.4E5
*OUTPUT, FIELD, VARIABLE=PRESELECT
*OUTPUT, HISTORY, VARIABLE=PRESELECT, FILTER=ANTIALIASING
*NODE OUTPUT, NSET=REFPUNCH
```

```
 RF2, U2
*END STEP
```

For the second explicit dynamics step set the time period to **0.007** s. Delete the *CONTACT CONTROLS option (it is relevant only for Abaqus/Standard), and modify the boundary condition to use the amplitude curve **RAMP2**. The second step appears as follows:

```
**
** Step 2
**
*STEP
Apply punch stroke
*DYNAMIC, EXPLICIT
, 0.007
*BOUNDARY, AMPLITUDE=RAMP2
REFPUNCH,2,2,-0.030
*END STEP
```

To help determine how closely the analysis approximates the quasi-static assumption, the various energy histories will be useful. Especially useful is comparing the kinetic energy to the internal strain energy. The energy history is written to the output database file as part of the preselected history output.

Running the job

Save the input file and submit the job for analysis. Monitor the solution progress; correct any modeling errors that are detected, and investigate the cause of any warning messages.

Strategy for evaluating the results

Before looking at the results that are ultimately of interest, such as stresses and deformed shapes, we need to determine whether or not the solution is quasi-static. One good approach is to compare the kinetic energy history to the internal energy history. In a metal forming analysis most of the internal energy is due to plastic deformation. In this model the blank is the primary source of kinetic energy (the motion of the holder is negligible, and the punch and die have no mass associated with them). To determine whether an acceptable quasi-static solution has been obtained, the kinetic energy of the blank should be no greater than a few percent of its internal energy. For greater accuracy, especially when springback stresses are of interest, the kinetic energy should be lower. This approach is very useful because it applies to all types of metal forming processes and does not require any intuitive understanding of the stresses in the model; many forming processes may be too complex to permit an intuitive feel for the results.

While a good primary indication of the caliber of a quasi-static analysis, the ratio of kinetic energy to internal energy alone is not adequate to confirm the quality. You should also evaluate the two energies independently to determine whether they are reasonable. This part of the evaluation

takes on increased importance when accurate springback stress results are needed because an accurate springback stress solution is highly dependent on accurate plasticity results. Even if the kinetic energy is fairly small, if it contains large oscillations, the model could be experiencing significant plasticity. Generally, we expect smooth loading to produce smooth results; if the loading is smooth but the energy results are oscillatory, the results may be inadequate. Since an energy ratio is incapable of showing such behavior, you should also study the kinetic energy history itself to see whether it is smooth or noisy.

If the kinetic energy does not indicate quasi-static behavior, it can be useful to look at velocity histories at some nodes to get an understanding of the model's behavior in various regions. Such velocity histories can indicate which regions of the model are oscillating and causing the high kinetic energies.

Evaluating the results

Enter Abaqus/Viewer, and open the output database created by this job (**`channel_xpl.odb`**). Plot the whole model kinetic (**`ALLKE`**) and internal (**`ALLIE`**) energies.

History plots of the kinetic and internal energies for the whole model appear as shown in Figure 13–9 and Figure 13–10, respectively.

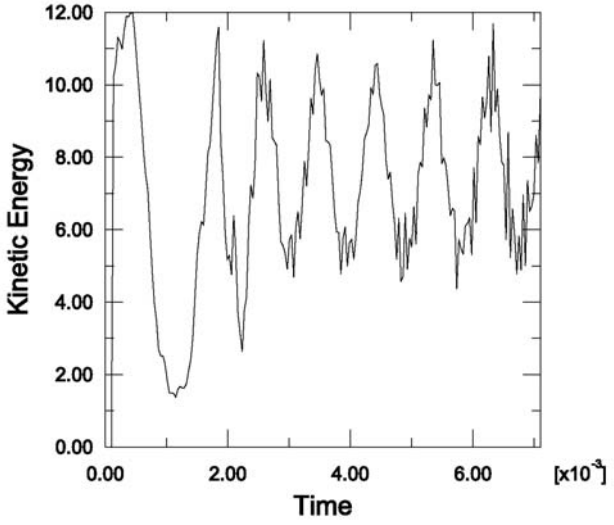


Figure 13–9 Kinetic energy history for forming analysis, attempt 1.

The kinetic energy history shown in Figure 13–9 oscillates significantly. In addition, the kinetic energy history has no clear relation to the forming of the blank, which indicates the inadequacy of this analysis. In this analysis the punch velocity remains constant, while the kinetic energy—which is primarily due to the motion of the blank—is far from constant.

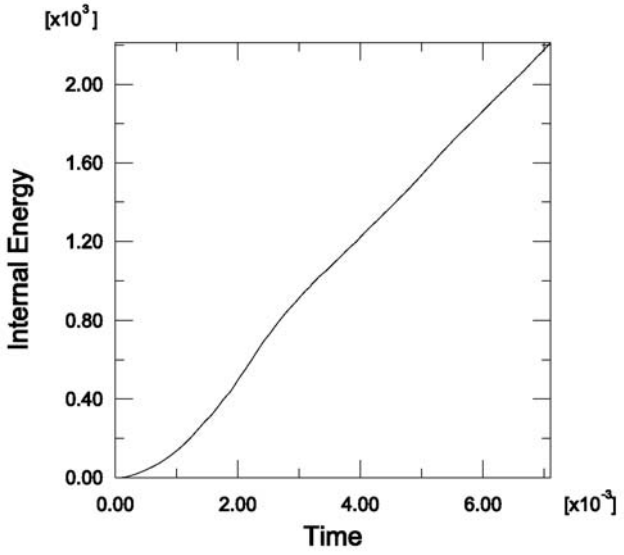


Figure 13–10 Internal energy history for forming analysis, attempt 1.

Comparing Figure 13–9 and Figure 13–10 shows that the kinetic energy is a small fraction (less than 1%) of the internal energy through all but the very beginning of the analysis. The criterion that kinetic energy must be small relative to internal energy has been satisfied, even for this severe loading case.

Although the kinetic energy of the model is a small fraction of the internal energy, it is still quite noisy. Therefore, we should change the simulation in some way to obtain a smoother response.

13.5.2 Forming analysis—attempt 2

Even if the punch actually moves at a nearly constant velocity, the results of the first simulation attempt indicate it is desirable to use a different amplitude curve that allows the blank to accelerate more smoothly. When considering what type of loading amplitude to use, remember that smoothness is important in all aspects of a quasi-static analysis. The preferred approach is to move the punch as smoothly as possible the desired distance in the desired amount of time.

We will now analyze the forming stage using a smoothly applied holder force and a smoothly applied punch displacement; we will compare the results to those obtained earlier. Refer to "Smooth amplitude curves," Section 13.2.1, for an explanation of the smooth step amplitude curve.

Revise the existing amplitude curve definitions **`RAMP1`** and **`RAMP2`** so that they define smooth step amplitude curves. You can make this change by appending DEFINITION=SMOOTH STEP to each ∗AMPLITUDE option. The amplitude curves will now be smooth in both their first and second

derivatives. Therefore, using a smooth step amplitude curve for the displacement control also assures us that the velocity and acceleration are smooth.

Save the **`channel_xpl.inp`** input file and submit it for analysis. Monitor the solution progress; correct any modeling errors that are detected, and investigate the cause of any warning messages.

Evaluating the results for attempt 2

The kinetic energy history is shown in Figure 13–11.

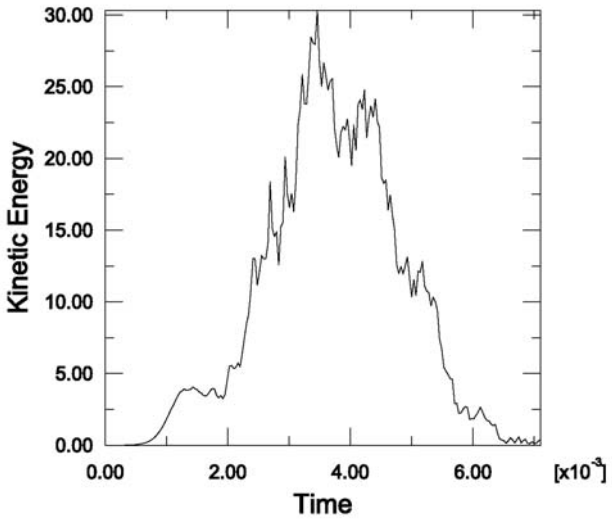


Figure 13–11 Kinetic energy history for forming analysis, attempt 2.

The response of the kinetic energy is clearly related to the forming of the blank: the value of kinetic energy peaks in the middle of the second step, corresponding to the time when the punch velocity is the greatest. Thus, the kinetic energy is appropriate and reasonable.

The internal energy for attempt 2, shown in Figure 13–12, shows a smooth increase from zero up to the final value. Again, the ratio of kinetic energy to internal energy is quite small and appears to be acceptable. Figure 13–13 compares the internal energy in the two forming attempts.

13.5.3 Discussion of the two forming attempts

Our initial criteria for evaluating the acceptability of the results was that the kinetic energy should be small compared to the internal energy. What we found was that even for the most severe case, attempt 1, this condition seems to have been met adequately. The addition of a smooth step amplitude curve helped reduce the oscillations in the kinetic energy, yielding a satisfactory quasi-static response.

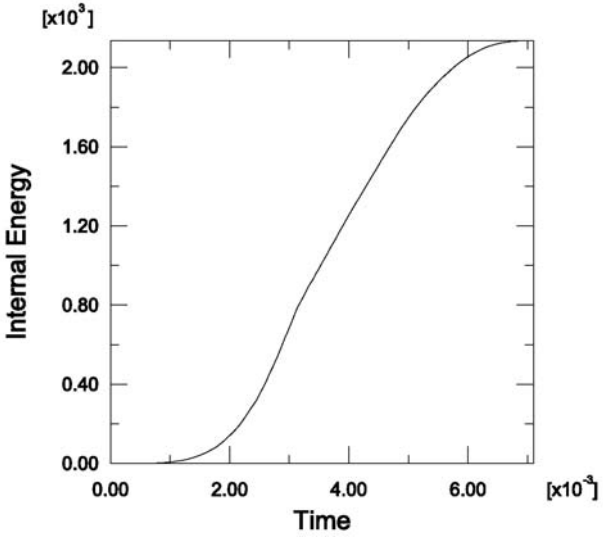


Figure 13–12 Internal energy history for forming analysis, attempt 2.

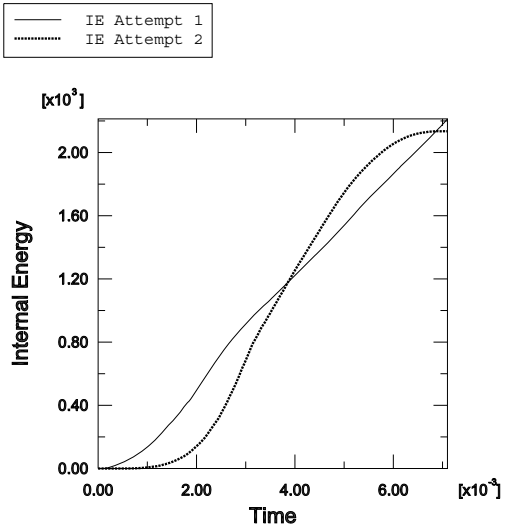


Figure 13–13 Comparison of internal energies for the two attempts of the forming analysis.

The additional requirements—that the histories of kinetic energy and internal energy must be appropriate and reasonable—are very useful and necessary, but they also increase the subjectivity of

evaluating the results. Enforcing these requirements in general for more complex forming processes may be difficult because these requirements demand some intuition regarding the behavior of the forming process.

Results of the forming analysis

Now that we are satisfied that the quasi-static solution for the forming analysis is adequate, we can study some of the other results of interest. Figure 13–14 shows a comparison of the Mises stress in the blank obtained with Abaqus/Standard and Abaqus/Explicit.

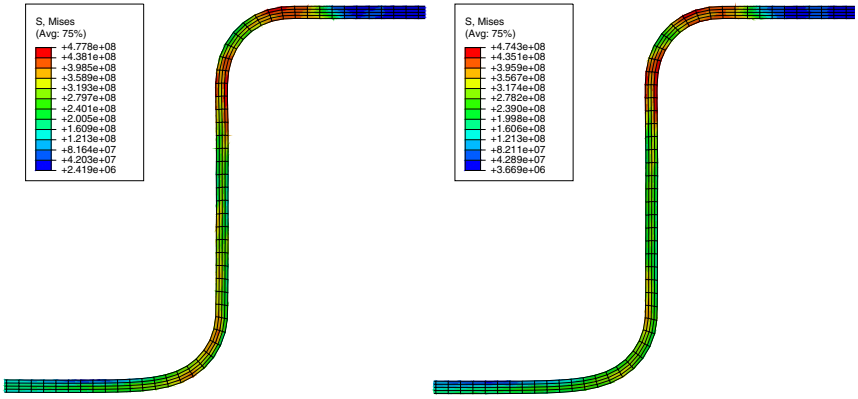


Figure 13–14 Contour plot of Mises stress in Abaqus/Standard (left) and Abaqus/Explicit (right) channel forming analyses.

The plot shows that the peak stresses in the Abaqus/Standard and Abaqus/Explicit analyses are within 1% of each other and that the overall stress contours of the blank are very similar. To further examine the validity of the quasi-static analysis results, you should compare the equivalent plastic strain results and final deformed shapes from the two analyses.

Figure 13–15 shows contour plots of the equivalent plastic strain in the blank, and Figure 13–16 shows an overlay plot of the final deformed shape predicted by the two analyses. The equivalent plastic strain results for the Abaqus/Standard and Abaqus/Explicit analyses are within 5% of each other. In addition, the final deformed shape comparison shows that the explicit quasi-static analysis results are in excellent agreement with the results from the Abaqus/Standard static analysis.

You should also compare the steady punch force predicted by the Abaqus/Standard and Abaqus/Explicit analyses.

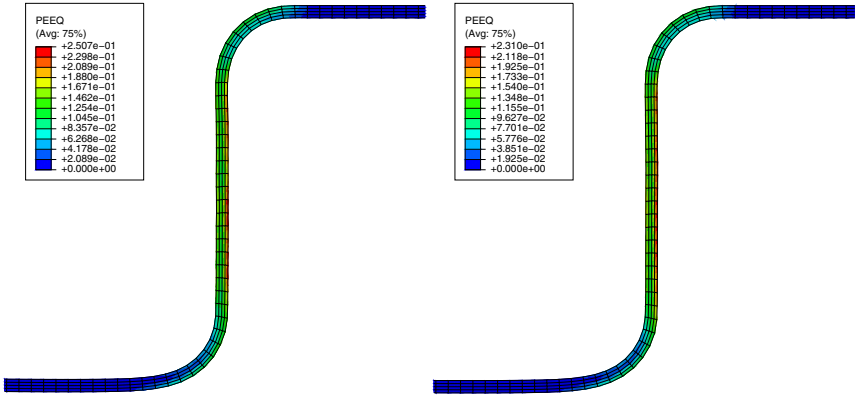


Figure 13–15 Contour plot of PEEQ in Abaqus/Standard (left) and Abaqus/Explicit (right) channel forming analyses.

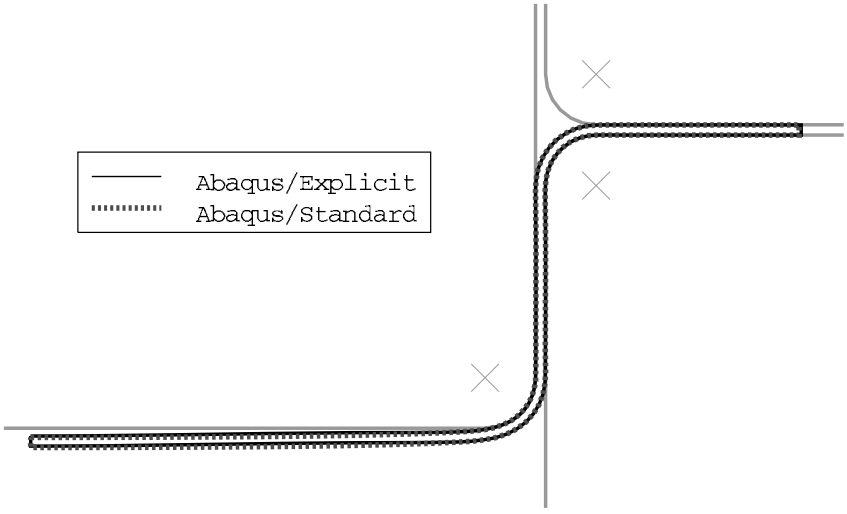


Figure 13–16 Final deformed shape in Abaqus/Standard and Abaqus/Explicit forming analyses.

To compare the punch force-displacement histories:

1. Save the punch displacement (**U2**) and reaction force (**RF2**) history data from the Abaqus/Standard analysis as **U2-std** and **RF2-std**, respectively. The number of the punch reference node is **7000**.
2. Similarly, save punch displacement (**U2**) and reaction force (**RF2**) history data from the Abaqus/Explicit analysis as **U2-xpl** and **RF2-xpl**, respectively.

   Next, you will operate on saved *X–Y* data to create the force-displacement curves. In the force-displacement plot we would like the downward motion of the punch to be represented as a positive value; therefore, when you create the force-displacement curves include a negative sign before the displacement history data so that motion in the negative 2-direction will be positive.
3. In the Results Tree, double-click **XYData**; then select **Operate on XY data** in the **Create XY Data** dialog box. Click **Continue**.
4. In the **Operate on XY Data** dialog box, combine the force and displacement history data from the Abaqus/Standard analysis to create a force-displacement curve. The expression at the top of the dialog box should appear as:

   ```
   combine ( -"U2-std", "RF2-std" )
   ```
5. Click **Save As** to save the calculated displacement curve as **forceDisp-std**.
6. In the **Operate on XY Data** dialog box, combine the force and displacement history data from the Abaqus/Explicit analysis to create a force-displacement curve. The expression at the top of the dialog box should appear as:

   ```
   combine ( -"U2-xpl", "RF2-xpl" )
   ```
7. Click **Save As** to save the calculated displacement curve as **forceDisp-xpl**.
8. Plot **forceDisp-std** and **forceDisp-xpl** in the viewport.

   There is significantly more noise in the Abaqus/Explicit results compared to the Abaqus/Standard results because Abaqus/Explicit simulates a quasi-static response while Abaqus/Standard solves for true static equilibrium. Some of the noise in the Abaqus/Explicit history data was removed during the analysis by the built-in anti-aliasing filter specified on the output request. Now, you will use an Abaqus/Viewer *X–Y* data filter to remove more of the solution noise from the Abaqus/Explicit force-displacement curve. The Abaqus/Viewer *X–Y* data filters should only be applied to *X–Y* data whose *X*-value is time. This avoids confusion regarding the meaning of the filter cutoff frequency and prevents problems with the data regularization that is performed internally before the filter is applied. Consequently, you will not filter **forceDisp-xpl** directly, but rather you will filter **U2-xpl** and **RF2-xpl** individually before combining them to create a new force-displacement curve. It is best to apply the same filter operations (both during the analysis and during postprocessing) to any two *X–Y* data objects that will be combined. This will ensure that any distortions due to filtering (such as time delays) are uniformly applied to the combined data.

9. In the **Operate on XY Data** dialog box, filter the force history data using a Butterworth filter with a cutoff frequency of 1100 Hz. The expression at the top of the dialog box should appear as:

   ```
   butterworthFilter(xyData="RF2-xpl",cutoffFrequency=1100)
   ```

   Note: Choosing an appropriate filter cutoff frequency takes engineering judgment and a good understanding of the physical system being modeled. Often an iterative approach (beginning with a relatively high cutoff frequency and then gradually reducing it) can be used to find a cutoff frequency that removes solution noise with minimal distortion of the underlying physical solution. Knowledge of the system's natural frequencies can also assist in the determination of appropriate filter cutoff frequencies. For this example, we performed a frequency extraction analysis to determine the fundamental frequency of the undeformed blank (140 Hz); however, the blank at the end of the forming step will have a fundamental frequency that is considerably higher. If you perform a natural frequency extraction analysis on the final model configuration, you will find that the fundamental frequency at the end of the forming step is approximately 1000 Hz. Hence, a cutoff frequency that is slightly larger than this value is a good choice for this model.

10. Click **Save As** to save the calculated displacement curve as **RF2-xpl-bw1100**.

11. Similarly, filter the displacement history data using a Butterworth filter with a cutoff frequency of 1100 Hz. The expression at the top of the **Operate on XY Data** dialog box should appear as:

    ```
    butterworthFilter(xyData="U2-xpl",cutoffFrequency=1100)
    ```

12. Click **Save As** to save the calculated displacement curve as **U2-xpl-bw1100**.

13. Combine the filtered Abaqus/Explicit force and displacement histories. The expression at the top of the **Operate on XY Data** dialog box should appear as:

    ```
    combine ( -"U2-xpl-bw1100", "RF2-xpl-bw1100" )
    ```

14. Click **Save As** to save the calculated displacement curve as **forceDisp-xpl-bw1100**.

15. Add **forceDisp-xpl-bw1100** to the plot of **forceDisp-std** and **forceDisp-xpl**. Customize the plot appearance to obtain a plot similar to Figure 13–17.

As seen in Figure 13–17, the steady punch force predicted by Abaqus/Explicit is approximately 12% higher than that predicted by Abaqus/Standard. The differences between the Abaqus/Standard and Abaqus/Explicit results are primarily due to two factors. First, Abaqus/Explicit regularizes the material data. Second, friction effects are handled slightly differently in the two analysis products; Abaqus/Standard uses penalty friction, whereas Abaqus/Explicit uses kinematic friction.

From these comparisons it is clear that both Abaqus/Standard and Abaqus/Explicit are capable of handling difficult contact analyses such as this one. However, there are some advantages to running this type of analysis in Abaqus/Explicit: Abaqus/Explicit is able to handle complex contact conditions more

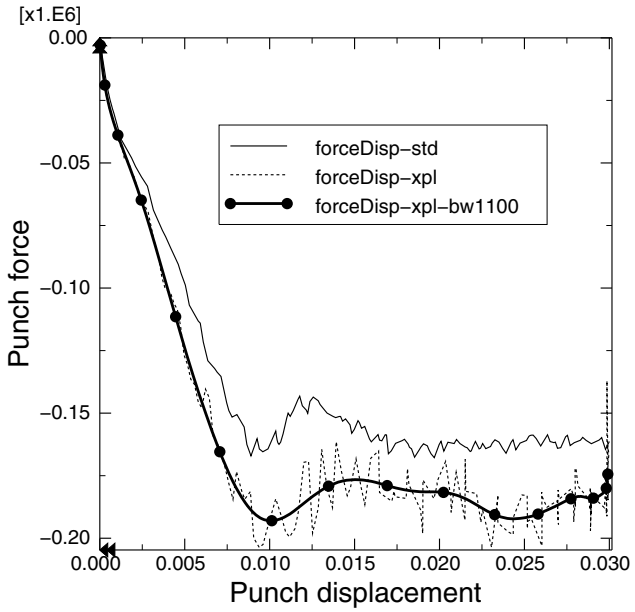


Figure 13–17 Steady punch force comparison for Abaqus/Standard and Abaqus/Explicit.

readily. However, when choosing Abaqus/Explicit for quasi-static analysis, you should be aware that you may need to iterate on an appropriate loading rate. In determining the loading rate, it is recommended that you begin with faster loading rates and decrease the loading rate as necessary. This will help optimize the run time for the analysis.

13.5.4 Methods of speeding up the analysis

Now that we have obtained an acceptable solution to the forming analysis, we can try to obtain similar acceptable results using less computer time. Most forming analyses require too much computer time to be run in their physical time scale because the actual time period of forming events is large by explicit dynamics standards; running in an acceptable amount of computer time often requires making changes to the analysis to reduce the computer cost. There are two ways to reduce the cost of the analysis:

1. Artificially increase the punch velocity so that the same forming process occurs in a shorter step time. This method is called *load rate scaling*.
2. Artificially increase the mass density of the elements so that the stability limit increases, allowing the analysis to take fewer increments. This method is called *mass scaling*.

Unless the model has rate-dependent materials or damping, these two methods effectively do the same thing.

Determining acceptable mass scaling

"Loading rates," Section 13.2, and "Metal forming problems," Section 13.2.3, discuss how to determine acceptable scaling of the loading rate or mass to reduce the run time of a quasi-static analysis. The goal is to model the process in the shortest time period in which inertial forces remain insignificant. There are bounds on how much scaling can be used while still obtaining a meaningful quasi-static solution.

As discussed in "Loading rates," Section 13.2, we can use the same methods to determine an appropriate mass scaling factor as we would use to determine an appropriate load rate scaling factor. The difference between the two methods is that a load rate scaling factor of f has the same effect as a mass scaling factor of f^2. Originally, we assumed that a step time on the order of the period of the fundamental frequency of the blank would be adequate to produce quasi-static results. By studying the model energies and other results, we were satisfied that these results were acceptable. This technique produced a punch velocity of approximately 4.3 m/s. Now we will accelerate the solution time using mass scaling and compare the results against our unscaled solution to determine whether the scaled results are acceptable. We assume that scaling can only diminish, not improve, the quality of the results. The objective is to use scaling to decrease the computer time and still produce acceptable results.

Our goal is to determine what scaling values still produce acceptable results and at what point the scaled results become unacceptable. To see the effects of both acceptable and unacceptable scaling factors, we investigate a range of scaling factors on the stable time increment size from $\sqrt{5}$ to 5; specifically, we choose $\sqrt{5}$, $\sqrt{10}$, and 5. These speedup factors translate into mass scaling factors of 5, 10, and 25, respectively.

To apply mass scaling of 5, add the following option to the history definition,

```
*FIXED MASS SCALING, ELSET=BLANK, FACTOR=5
```

and save the modified input in a file named **channel_xpl_5.inp** and submit it for analysis; also create input files named **channel_xpl_10.inp** and **channel_xpl_25.inp** with mass scaling factors of 10 and 25, respectively, and submit them for analysis.

First, we will look at the effect of mass scaling on the equivalent plastic strains and the displaced shape. We will then see whether the energy histories provide a general indication of the analysis quality.

Evaluating the results with mass scaling

One of the results of interest in this analysis is the equivalent plastic strain, PEEQ. Since we have already seen the contour plot of PEEQ at the completion of the unscaled analysis in Figure 13–15, we can compare the results from each of the scaled analyses with the unscaled analysis results. Figure 13–18 shows PEEQ for a speedup of $\sqrt{5}$ (mass scaling of 5), Figure 13–19 shows PEEQ for a speedup of $\sqrt{10}$ (mass scaling of 10), and Figure 13–20 shows PEEQ for a speedup of 5 (mass scaling of 25). Figure 13–21 compares the internal and kinetic energy histories for each case of mass

scaling. The mass scaling case using a factor of 5 yields results that are not significantly affected by the increased loading rate. The case with a mass scaling factor of 10 shows a high kinetic-to-internal energy ratio, yet the results seem reasonable when compared to those obtained with slower loading rates. Thus, this is likely close to the limit on how much this analysis can be sped up. The final case, with a mass scaling factor of 25, shows evidence of strong dynamic effects: the kinetic-to-internal energy ratio is quite high, and a comparison of the final deformed shapes among the three cases demonstrates that the deformed shape is significantly affected in the last case.

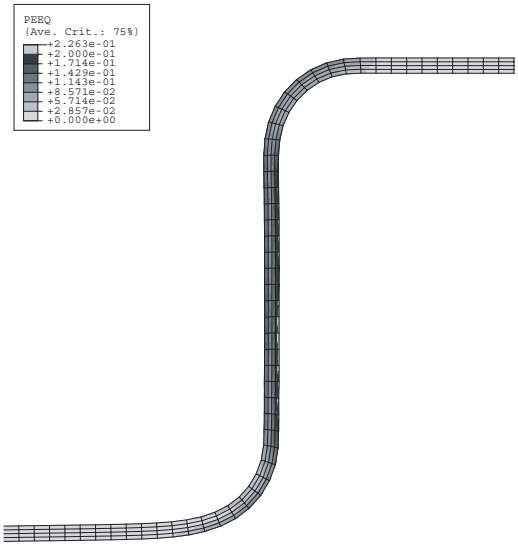


Figure 13–18 Equivalent plastic strain PEEQ for speedup of $\sqrt{5}$ (mass scaling of 5).

Discussion of speedup methods

As the mass scaling increases, the solution time decreases. The quality of the results also decreases because dynamic effects become more prominent, but there is usually some level of scaling that improves the solution time without sacrificing the quality of the results. Clearly, a speedup of 5 is too great to produce quasi-static results for this analysis.

A smaller speedup, such as $\sqrt{5}$, does not alter the results significantly. These results would be adequate for most applications, including springback analyses. With a scaling factor of 10 the quality of the results begins to diminish, while the general magnitudes and trends of the results remain intact. Correspondingly, the ratio of kinetic energy to internal energy increases noticeably. The results for this case would be adequate for many applications but not for accurate springback analysis.

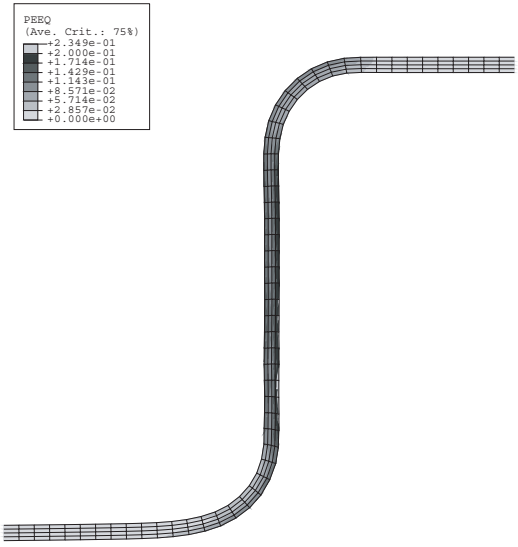


Figure 13–19 Equivalent plastic strain PEEQ for speedup of $\sqrt{10}$ (mass scaling of 10).

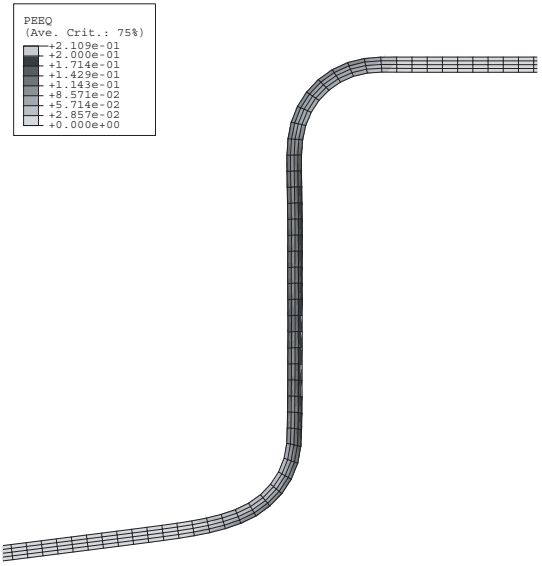


Figure 13–20 Equivalent plastic strain PEEQ for speedup of 5 (mass scaling of 25).

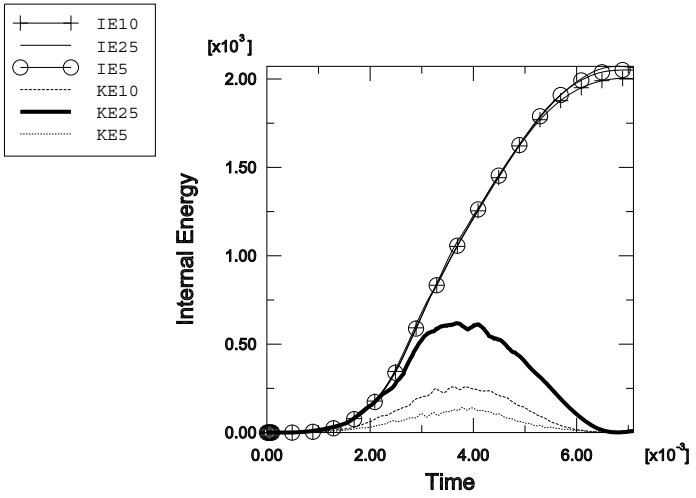


Figure 13–21 Kinetic and internal energy histories for mass scaling factors of 5, 10, and 25, corresponding to speedup factors of $\sqrt{5}$, $\sqrt{10}$, and 5, respectively.

13.5.5 Springback analysis in Abaqus/Standard

While it is possible to perform springback analyses within Abaqus/Explicit, Abaqus/Standard is much more efficient at solving springback analyses. Since springback analyses are simply static simulations without external loading or contact, Abaqus/Standard can obtain a springback solution in just a few increments. Conversely, Abaqus/Explicit must obtain a dynamic solution over a time period that is long enough for the solution to reach a steady state. For efficiency Abaqus has the capability to transfer results back and forth between Abaqus/Explicit and Abaqus/Standard, allowing you to perform forming analyses in Abaqus/Explicit and springback analyses in Abaqus/Standard.

The steps that follow assumes that you have access to the full input file for this example. This input file, **channel_springback.inp**, is provided in "Forming a channel with Abaqus/Standard," Section A.13, in the online HTML version of this manual. Instructions on how to fetch and run the script are given in Appendix A, "Example Files."

The first option following *HEADING is the *IMPORT option, which reads the element definitions and the state from the corresponding Abaqus/Explicit analysis. This import input file begins with the following:

```
*HEADING
 Analysis of the forming of a channel -- springback
 Abaqus/Standard springback following channel_xpl_5.inp
 SI units (kg, m, s, N)
```

```
*IMPORT, STEP=2, STATE=YES, UPDATE=NO
BLANK,
*IMPORT NSET
CENTER, MIDLEFT
```

Setting the STATE parameter equal to YES causes the state of the model—stresses, strains, etc.—to be imported. Setting the UPDATE parameter equal to NO causes the strains and displacements to be imported as well instead of being reset to zero. The data line following the *IMPORT option supplies the name of the element set containing the elements that are to be imported. The *IMPORT NSET option identifies node set names to be imported.

Next, create a general static step. Set the initial time increment to **0.1**, and include the effects of geometric nonlinearity (note that the Abaqus/Explicit analysis considered them; this is the default setting in Abaqus/Explicit). Springback analyses can suffer from instabilities that adversely affect convergence. Thus, include automatic stabilization to prevent this problem. Use the default value for the dissipated energy fraction.

You must redefine the boundary conditions, which are not imported. Impose the same XSYMM-type displacement boundary conditions that were imposed in the Abaqus/Explicit model on the set **Center**.

To remove rigid body motion, it is necessary to fix a single point in the blank, such as set **MidLeft**, in the 2-direction (in this way you impose no unnecessary constraints). Rather than apply a displacement boundary condition to this point, apply a zero-velocity boundary condition to fix this point at its final position at the end of the forming stage. This will allow the model to retain continuity in the blank location through any additional forming stages that may follow.

The complete history definition is as follows:

```
*STEP, NLGEOM=YES
*STATIC, STABILIZE, ALLSDTOL=0
0.1, 1.
*BOUNDARY
CENTER, XSYMM
*BOUNDARY, TYPE=VELOCITY
MIDLEFT, 2, 2
*END STEP
```

Save your input in a file named **channel_springback.inp**, and run the analysis using the following command:

```
abaqus job=channel_springback oldjob=channel_xpl_5
```

Results of the springback analysis

Figure 13–22 overlays (**View→Overlay Plot**) the deformed shape of the blank after the forming and springback stages (the forming stage corresponds to the last frame of the Abaqus/Explicit output database file, while the springback stage corresponds to the final frame of the Abaqus/Standard

output database file). The springback result is necessarily dependent on the accuracy of the forming stage preceding it. In fact, springback results are highly sensitive to errors in the forming stage, more sensitive than the results of the forming stage itself.

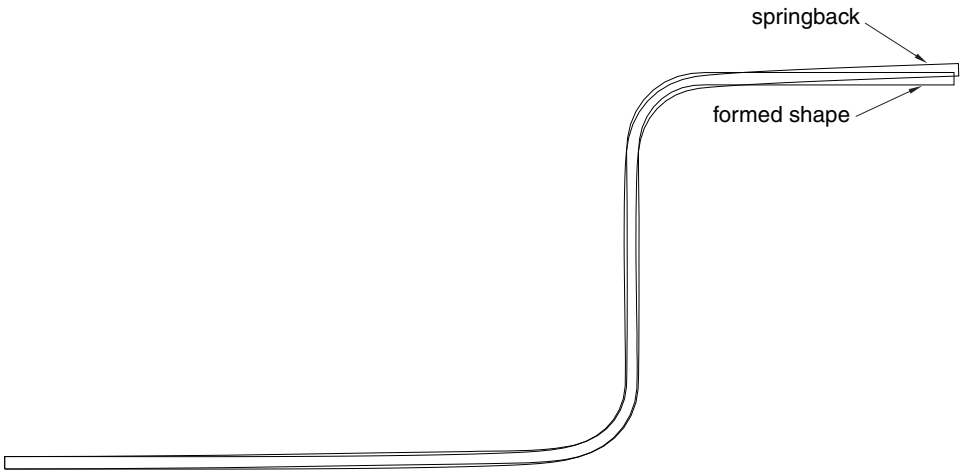


Figure 13–22 Deformed model shapes following forming and springback.

You should also plot the blank's internal energy ALLIE and compare it with the static stabilization energy ALLSD that is dissipated. The stabilization energy should be a small fraction of the internal energy to have confidence in the results. Figure 13–23 shows a plot of these two energies; the static stabilization energy is indeed small and, thus, has not significantly affected the results.

13.6 Summary

- If a quasi-static analysis is performed in its natural time scale, the solution should be nearly the same as a truly static solution.
- It is often necessary to use load rate scaling or mass scaling to obtain a quasi-static solution using less CPU time.
- The loading rate often can be increased somewhat, as long as the solution does not localize. If the loading rate is increased too much, inertial forces adversely affect the solution.
- Mass scaling is an alternative to increasing the loading rate. When using rate-dependent materials, mass scaling is preferable because increasing the loading rate artificially changes the material properties.

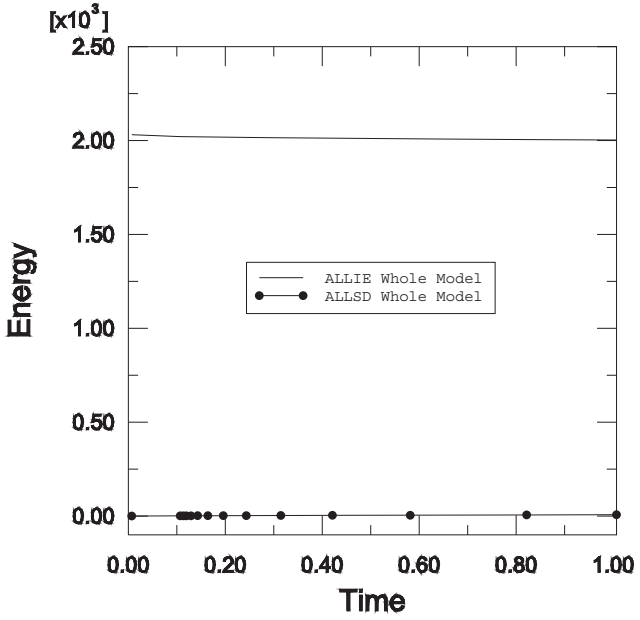


Figure 13–23 Internal and static stabilization energy histories.

- In a static analysis the lowest modes of the structure dominate the response. Knowing the lowest natural frequency and, correspondingly, the period of the lowest mode, you can estimate the time required to obtain the proper static response.
- It may be necessary to run a series of analyses at varying loading rates to determine an acceptable loading rate.
- The kinetic energy of the deforming material should not exceed a small fraction (typically 5% to 10%) of the internal energy throughout most of the simulation.
- Using the *AMPLITUDE option with the DEFINITION=SMOOTH STEP parameter is the most efficient way to prescribe displacements in a quasi-static analysis.
- Import the model from Abaqus/Explicit to Abaqus/Standard to perform an efficient springback analysis.

Appendix A: Example Files

This appendix contains a list of complete input files for the examples contained in this guide. You can get a copy of any of these input files with the command

abaqus fetch job=*file_name*

where *file_name* does not include the extension **.inp**.

A.1 Overhead hoist frame

- frame.inp
- frame_xpl.inp

A.2 Connecting lug

- lug.inp
- lug_xpl.inp

A.3 Skew plate

- skew.inp

A.4 Cargo crane

- crane.inp

A.5 Cargo crane – dynamic loading

- dynamics.inp
- dynamics_xpl.inp

A.6 Nonlinear skew plate

- skew_nl.inp
- skew_nl_xpl.inp

A.7 Stress wave propagation in a bar

- wave_50x10x10.inp
- wave_25x5x5.inp

- wave_50x5x5.inp
- wave_50x10x5.inp

A.8 Connecting lug with plasticity

- lug_plas.inp
- lug_plas_hard.inp

A.9 Blast loading on a stiffened plate

- blast_base.inp
- blast_damp.inp
- blast_long.inp
- blast_refined.inp
- blast_rate.inp

A.10 Axisymmetric mount

- mount.inp

A.11 Test fit of hyperelastic material data

- single_elem2.inp

A.12 Vibration of a piping system

- pipe.inp
- pipe-2.inp

A.13 Forming a channel with Abaqus/Standard

- channel.inp

A.14 Shearing of a lap joint

- lap_joint.inp

A.15 Circuit board drop test

- circuit.inp

A.16 Forming a channel with Abaqus/Explicit

- channel_freq.inp
- channel_xpl.inp
- channel_xpl_5.inp
- channel_xpl_10.inp
- channel_xpl_25.inp
- channel_springback.inp

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis; multiphysics solutions for insight into challenging engineering problems; and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Vélizy, France, SIMULIA provides sales, services, and support through a global network of regional offices and distributors. For more information, visit www.simulia.com.

About Dassault Systèmes

As a world leader in 3D and Product Lifecycle Management (PLM) solutions, Dassault Systèmes brings value to more than 100,000 customers in 80 countries. A pioneer in the 3D software market since 1981, Dassault Systèmes develops and markets PLM application software and services that support industrial processes and provide a 3D vision of the entire lifecycle of products from conception to maintenance to recycling. The Dassault Systèmes portfolio consists of CATIA for designing the virtual product, SolidWorks for 3D mechanical design, DELMIA for virtual production, SIMULIA for virtual testing, ENOVIA for global collaborative lifecycle management, and 3DVIA for online 3D lifelike experiences. Dassault Systèmes' shares are listed on Euronext Paris (#13065, DSY.PA), and Dassault Systèmes' ADRs may be traded on the US Over-The-Counter (OTC) market (DASTY). For more information, visit www.3ds.com.

www.3ds.com Abaqus, the 3DS logo, SIMULIA, CATIA, SolidWorks, DELMIA, ENOVIA, 3DVIA, and Unified FEA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries. Other company, product, and service names may be trademarks or service marks of their respective owners. © Dassault Systèmes, 2012

DASSAULT SYSTEMES